eta/VPG
USER'S PROCESSOR MANUAL

An LS-DYNA Based Full-Vehicle

Simulation Solution Package.

VPG Version: 1.1
Manual Release Date: June 15, 2000

FORWARD

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design.

This material is a compilation of data and figures from many sources.

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Engineering Technology Associates, Inc.
1133 East Maple Road, Suite 200
Troy, MI 48083

Phone: (248) 729-3010
Support: (800) ETA-3362
Fax: (248) 729-3020

Engineering Technology Associates, Inc., ETA, the ETA logo, and eta/VPG are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of the respective owners.

Copyright 2000 Engineering Technology Associates, Inc. All rights reserved

TABLE OF CONTENTS

INTRODUCTION ..... Section 1
- Documentation ..... Section 1.1
- eta/VPG History ..... Section 1.2
- eta/VPG Methodology ..... Section 1.3
- eta/VPG Software Package ..... Section 1.4
- eta/VPG Special Features ..... Section 1.5

FEATURES AND FUNCTIONS OVERVIEW ..... Section 2
- General ..... Section 2.0
- Pre Processor ..... Section 2.1
- Post Processor ..... Section 2.2
- Menu System ..... Section 2.3
- Function Keys ..... Section 2.4
- Display Window ..... Section 2.5
- Mouse Functions ..... Section 2.6
- Keyboard Entry ..... Section 2.7
- Specifications ..... Section 2.8
- Line Data ..... Section 2.9
- Conventions ..... Section 2.10
- VPG File Menu ..... Section 2.11
- Recommended Naming Convention ..... Section 2.12
- LS-DYNA Cards ..... Section 2.13
- Local Coordinate System ..... Section 2.14
- Entity Selction ..... Section 2.15

GETTING STARTED ..... Section 3
- Opening/Creating an eta/VPG Database File ..... Section 3.1
- Setting Up a VPG Model ..... Section 3.2

MAIN MENU ..... Section 4

FILE MANAGER ..... Section 5

PRE-PROCESSOR ..... Section 6
- LINE ..... Section 6.1
- SURFACE OPTIONS ..... Section 6.2
- ELEMENT OPTIONS ..... Section 6.3

NODE OPTIONS ..... Section 6.4
MODEL CHECKER ..... Section 6.5
SET MENU (LS-DYNA) ..... Section 6.6
SET MENU (NASTRAN) ..... Section 6.7
BOUNDARY CONDITIONS (LS-DYNA) ..... Section 6.8
BOUNDARY CONDITIONS (NASTRAN) ..... Section 6.9
MATERIAL PROPERTY (LS-DYNA) ..... Section 6.10
MATERIAL PROPERTY (NASTRAN) ..... Section 6.11
ELEMENT PROPERTY ..... Section 6.12
CONTACT INTERFACE ..... Section 6.13
DYNA MISCELLANEOUS ..... Section 6.14
SUPER ELEMENT ..... Section 6.15

VPG MODULES ..... Section 7
ROAD MENU ..... Section 7.1
SUSPENSION MENU ..... Section 7.2
TIRE MODEL ..... Section 7.3

ANALYSIS EXECUTION ..... Section 8
RUN ANALYSIS ..... Section 8.1
RESTART ..... Section 8.2

POST PROCESSING ..... Section 9
FATIGUE ..... Section 9.1
POST PROCESSING ..... Section 9.2
GRAPH ..... Section 9.3

UTILITY ..... Section 10
VIEWING OPTIONS ..... Section 10.1
VIEWING OPTIONS WINDOW ..... Section 10.2
PART CONTROL ..... Section 10.3
SETUP DEFAULTS ..... Section 10.4
UTILITY ..... Section 10.5
DISPLAY PARAMETER OPTIONS WINDOW ..... Section 10.6

APPENDIX .....
VPG Capabilities for F.E.A. Analysis Programs ..... A
VPG Hardware and Software Requirements ..... B
Supported IGES Entity Types C

Converting RADIOSS to LS-DYNA ..... D
Converting LS-DYNA to RADIOSS ..... E
Converting NASTRAN to LS-DYNA ..... F
Converting LS-DYNA to NASTRAN ..... G
Converting RADIOSS to NASTRAN ..... H
Converting NASTRAN to RADIOSS I
Converting ADAMS to LS-DYNA ..... J
VPG Menu ..... K

FUNCTION INDEX .....

Engineering Technology Associates, Inc.
Creation Date: December 15, 1998
Revision Level: 5
Revision Date: November 2, 2000
Approved by: Bruce Morse

Introduction

1.1 DOCUMENTATION

The User's Manual consists of two volumes of documentation for the eta/Virtual Proving Ground (eta/VPG) software package; one for the Solver and one for the Processor. This manual documents the pre- and post-processing portion of the eta/VPG integrated program which was created to complete the following objectives:

1. To provide a general purpose pre- and post-processor for constructing FEA models and reviewing analysis results.
2. To provide utility modules for meeting VPG specific application requirements such as a tire model generator, a road surface library, a suspension model library, fatigue life prediction program and FFT post-processor.
3. To allow the user to create and retain user-defined modules and libraries for road surfaces, material models, fatigue life prediction modules, control modules, etc.
4. To interface with the solver, a specialized, double precision, version of the LS/DYNA Explicit and Implicit code.

Please reference the eta/VPG User's Manual - Solver, an integral part of the User's Manual, for analysis related usage. The eta/VPG User's Manual - Solver is a version of the LS/DYNA User's Manual Keyword Format (versions later than LS-940 Release).

The eta/VPG Applications Manual documents applications of the features and functions included in the User's Manual. Examples are provided to demonstrate the different techniques used in creating a VPG simulation. The Applications Manual also serves as a Training Manual for new users to gain experience with VPG techniques, methodology and code.

1.2 eta/VPG HISTORY

eta/VPG evolved in the mid 90's with the advent of faster computers, cost effective hardware, advanced software technology, the ability to perform event-based simulations and the need to overcome the limitations of existing CAE methodology. Conventional CAE techniques required smaller, specific analyses tailored to a specific task (such as NVH) and fractured from the global analysis of the automotive system. The eta/VPG developers were faced with the need for an integrated analysis that included both component level and full vehicle applications with the ability to perform real-time simulations. The analysis would be dynamic and nonlinear unlike the static, linear analysis techniques utilized by existing CAE practices.

ETA first tested this concept on a simple mechanism simulating an engine connecting rod, piston and crank shaft subsystem. The rigid body (piston and crankshaft) and flexible body (connecting rod) were combined for stress and strain evaluation of the connecting rod during a combustion event.

The engine combustion event was simulated as a dynamic, nonlinear event in real time and the results demonstrated that the motion and forces derived from the VPG simulation were the same as the rigid body linkage motion simulation results produced from a conventional rigid body linkage code. The benefit of obtaining realistic stress and strain results of the connecting rod from VPG simulation, in an event based fashion, produced valuable analysis data that could not be generated in traditional FEA approaches.

Additional studies of chassis/suspension mechanisms and/or linkage subsystems using flexible body approaches were performed and the results proved that the time domain solution was feasible. It also revealed that VPG's great potential was yet to be developed.

ETA engineers moved another step forward by simulating a rotating tire impact in a pothole event. A tire model was developed to include the mass flow rate technique to account for the tire air pressure variation in an impact condition. The simulation of the frictional contact of the tire model with the rigid road surface was extremely realistic and correlated with existing tire data.

At the same time, Arthur Tang had an opportunity to view a video of MGA Proving Ground, a full services vehicle proving ground located in Burlington, Wisconsin, owned and operated by MGA Research Corporation. He suggested that the road surface profiles could be digitized from the proving ground construction drawings in a CAD surface format. FEA mesh could be generated from the CAD surface data and produce a selection of generic road profiles representing the characteristics of the proving ground testing conditions.

Arthur Tang presented his ideas to Dr. Pat Miller at MGA Research Corporation and obtained his approval to use the MGA Proving Ground Road Profile to try the VPG approach. A tire model, including simulated tire pressure using the mass flow rate approach, was placed on an MGA FEA road surface and rotated across a pothole impacting the front leap of the pothole. The rotating motion of the tire matched existing event data; the virtual proving ground concept was established.

Since, 1995 ETA has moved VPG development work into high gear. Tim Palmer has led ETA's Applications Group in performing a number of serious full-vehicle VPG projects. VPG work has been performed for, Ssangyang W1, Mack Truck Cab-in-White, Daewoo V-100 and M-100, Chrysler CCV and Hyundai Precision SUV. ETA has also provided sup-

port to the Ford WIN126 pilot feasibility study project and the GM Small Car Division's effort to evaluate the VPG concept.

After two years of development and evaluation, it was confirmed that VPG technology could produce repeatable, reliable and correlatable analysis results. The level of confidence from users and correlation results lead to aggressive simulation use and requirements. As a result of such requirements, the enhancement of eta/VPG's current solver technology became necessary.

Dr. John Hallquist, at Livermore Software Technology Corporation (LSTC) accepted the challenge of implementing various features and functions in LS/DYNA to meet VPG requirements.

In mid 1997, ETA and LSTC evaluated all the facts related to VPG technology and decided to produce a software product, eta/VPG, coupling the existing LS/DYNA and eta/FEMB to form a niche application code for VPG technology.

In January of 1998, eta/VPG version 1.0 was released to ETA's Applications Group for use in final feasibility studies and production projects supporting Chrysler's CCV program, Ford's WIN126 program, and the GAZ 3111 program.

A VPG Users' Meeting was called in March 1998 to collect and better understand user requirements. Based on the user input, development goals and key features and functions of the code were finalized and eta/VPG Version 2.0 was ultimately released in August of 1998.

1.3 VPG METHODOLOGY

Virtual Proving Ground (VPG) is a general term used to reference a simulation methodology using the tire and proving ground road surface approach. VPG technology is also referenced as CAE Proving Ground, Analytical Proving Ground (by Ford users) and Virtual Road (by Chrysler users).

The VPG method is a set of techniques used with an explicit, nonlinear, dynamic analysis program which allows for the complete analysis of a mechanical system, including all joints, bushings, material and geometric non-linearities using an event based analysis. The class of problems targeted are those in which a mechanical system is to be analyzed in a dynamic sense. In other words, when a mechanical system is in use, the displacement, forces, accelerations and stresses occur in real time. The VPG method allows for the calculation of all of these quantities simultaneously, using a single analysis run.

While VPG methodology was initially developed for full vehicle simulations based on a proving ground durability cycle, the concept can also be applied to other dynamic mechanical systems.

1.4 eta/VPG SOFTWARE PACKAGE

eta/Virtual Proving Ground (eta/VPG) is a fully integrated, dynamic, nonlinear, finite element software package used to create, analyze, edit and visualize dynamic nonlinear engineering problems. The software includes an integrated preprocessor, post processor, and solver. It is a complete CAE tool for applying theory and engineering principals common in areas of mechanical and structural engineering. eta/VPG provides a single package for use in analysis of multi-body dynamics problems, linear static, nonlinear static, and dynamic nonlinear finite element analysis.

eta/VPG's strength lies in its ability to integrate problems that are treated as multi-discipline by other software packages. eta/VPG allows the user to combine multi-body dynamics problems with structural finite element analysis problems, providing real time kinematics or dynamics, as well as the stress or strain response of the structure in real time.

eta/VPG consists of three primary modules, the Pre processor, Post Processor and Solver modules. The Pre and Post Processor are general purpose, full feature and capability, and contain a set of VPG utility modules for efficient model generation and analysis. The Solver module is a double precision version of LS/DYNA, developed and marketed by Livermore Software Technology Corporation (LSTC). LS/DYNA has both explicit and implicit solvers that are seamlessly switched through a “restart function”.

eta/VPG’s multiple specialty features, designed specifically for full vehicle analysis, allow for easy modeling, analysis and post processing of the results in a user-friendly environment. Joints and bushings, finite element meshes, boundary conditions, materials, properties, suspension system components, tire and road surface libraries are all easily defined. The graphic user interface allows for the generation of these models as well as the animation and evaluation of the analysis results.

1.5 eta/VPG SPECIAL FEATURES

eta/VPG was designed with unique features specifically for virtual proving ground simula-

tions which do not exist in other general purpose pre- and post-processors. The development of these features and functions was necessary to satisfy the following VPG requirements:

1. Generation of complicated full vehicle/full system models and retention of complex non-linear material properties, contact definitions, etc.
2. Generation and retention of user defined libraries and modules such as a tire library, proving ground road surface library and driver control modules, etc.
3. Post-processing of huge amounts of analysis data (displacement, forces, acceleration, stress and strain) derived from time domain solution of multiple events and the conversion from time domain results into frequency domain results (frequency, mode shape and PSD, etc.).
4. Post-processing of stress and strain results of multiple events and the conducting fatigue life prediction.
5. Development of chassis/suspension models and conducting of post-processing to evaluate the chassis/suspension design characteristics.

The key modules and libraries are briefly described below:

LS-DYNA 940 Interface Module

eta/VPG incorporates a complete, direct LS-DYNA 940 interface. eta/VPG reads and writes all LS-DYNA 940 cards eliminating the need for text editing of the input deck. Create and retain all material nonlinear properties, contact definition and loading conditions.

Tire Model Generator

eta/VPG's tire model generator allows for easy construction of tire models. The tire model generation tool employs specific tire geometry and inflation pressure to automatically construct a three-dimensional finite element model. Tire models are used for both vehicle durability and NVH applications.

Suspension Model Generator

eta/VPG's sophisticated suspension model generator automates the FEA modeling of the most popular suspension types. Suspensions may be modeled using flexible, finite element representations or rigid members. Material properties of the suspension component can be specified using nonlinear stress-strain characteristics of the material. Included in eta/

VPG's suspension library are these suspension types:

McPherson H-Arm	***Trailing Arm***
McPherson A-Arm	***Hotchkiss***
SolidAxle	***Twist Beam***
Five Link	***Honda 5-Link***
Quadra Link	***Short Long Arm***

Road Surface Library

eta/VPG'S road surfaces form a full vehicle durability evaluation platform ready for kinematics and stress analyses of component, subsystem, and vehicular models. Road surfaces are generated using any 3D data or selected from the VPG library, which contains digitized models of MGA Proving Grounds such as:

Pothole Tracks	***Ripple Tracks***
Alternate Roll Surface	***Washboards***
Cobblestone Tracks	***Chatterstrip***
Body Twist Lane	***Paved surface***

Fatigue Analysis Program Module

eta/VPG's built in post processor automatically performs fatigue life analysis prediction to identify the key damage events and stress amplitudes. This data is then used to calculate the percentage of fatigue life remaining at the completion of the durability cycle. eta/VPG displays these results in easily read, combined fatigue life contour plots.

Signal Processing Module

eta/VPG performs full vehicle NVH studies on simulated proving ground surfaces. eta/VPG automatically converts time domain analysis results into frequency-domain via FFT (Fast Fourier Transform). eta/VPG determines both low and mid range frequencies up to 250Hz for operating mode shapes, frequencies, structural and airborne noise and frequency responses such as idle shake, rough road, power train and wheel unbalance.

2.0 GENERAL

The eta/VPG Processor has a complete graphic user interface that is operated on UNIX based workstations (W/S) including IBM, HP, SUN, DEC-Alpha and SGI platforms using various popular operating systems. The model generation, input file preparation, and results processing activities are all done in a W/S environment. The solution, either explicit or implicit, can be executed on both local W/S and/or remote server systems.

The graphic package requires PHIGS Library to operate. The program is organized as a "tree structure" and operated and controlled by a MOTIF Graphic User Interface (GUI). The operator can either activate the functions via mouse pick and/or key in the commands via keyboard entry. The function keys are also available for the operator to quickly move between main menus.

2.1 PRE PROCESSOR

eta/VPG's extensive Preprocessing capabilities contain all the functions necessary for expedient, high quality modeling. Users can read in data with VPG's CAD interface, build their model from scratch, read in an existing model, or a combination of the approaches.

CAD Interface

eta/VPG enables users to quickly read in geometry data in IGES format from any CAD system. Supported IGES entity types include:

Name	Type
Null Entity	0
Circular Arc Entity	100
Composite Curve Entity	102
Conic Arc Entity	104
Copious Data Entity	106
Plane Entity	108
Line Entity	110
Parametric Spline Curve Entity	112
Parametric Spline Surface Entity	114
Point Entity	116
Ruled Surface Entity	118
Surface of Revolution Entity	120
Tabulated Cylinder Entity	122
Transformation Matrix Entity	124

Continued on next page.

Rational B-Spline Curve Entity	126
Rational B-Spline Surface Entity	128
Offset Surface Entity	140
Boundary Entity	141
Curve on a Parametric Surface Entity	142
Bounded Surface Entity	143
Trimmed (Parametric) Surface Entity	144
Subfigure Definition Entity	308
Associativity Instance Entity	402
Property Entity	406
Singular Subfigure Instance Entity	408

For the German automotive industry, VDA file format is supported. Supported VDA entity types include:

POINT
CIRCLE
CURVE
SURF
CONS
FACE

VPG also supports line/surface data from PDGS and CGS via an external translator.

If CAD data is not available, eta/VPG has a complete geometry and surface building capability, which acts as an integrated CAD data generator.

Extensive Model Building Functions

eta/VPG includes a comprehensive selection of functions for creating and modifying line data and CAD surfaces. Element generation through 2-line, 4-line, 6-line and 8-line mesh creates beam, shell and solid elements.
-*Comprehensive Model Modification Functions*
-*Full System Assembly*

Automeshing

eta/VPG's automeshing function easily eliminates 90% of the time required to mesh trimmed and standard IGES surfaces. The automeshing function creates quadrilateral elements with a minimum of triangular elements.

Continued on next page.

Material/ Element Properties

eta/VPG supports all LS-DYNA material and element property cards. eta/VPG's ability to create and assign material and element properties directly on a displayed model greatly reduces the amount of editing required. Definition cards appear at specific junctions in the session prompting the user for material/ element properties. Users also have the option to specify an analysis code (NASTRAN, LS-DYNA) prior to assigning properties.

Contact Interface

eta/VPG seamlessly interfaces with LS-DYNA allowing the user to create and assign impact, sliding, or automatic interfaces for VPG applications. eta/VPG displays contact properties in easily read and modified CONTACT CARDS with a high degree of speed and flexibility.

Boundary Conditions

eta/VPG's BOUNDARY CONDITIONS menu allows the user to quickly create and verify constraints and loads on VPG models.

Extensive Model Integrity Checking Functions

The functions in eta/VPG's MODEL CHECKER menu quickly validate models for element orientation, size, skew, connectivity, and interior angles. Model validation default values are easily adjusted to suit the user's needs.

Constraints

eta/VPG supports all LS-DYNA Constraint Cards for quick, easy definition and manipulation of joints, welds, rivets, etc.

2.2 POST-PROCESSOR

VPG integrates a general purpose, full feature and capability post-processor for complete post processing within the single-code, workstation environment. Analysis results are seamlessly input from VPG's double precision LS-DYNA solver to the post processor for quick, easy interpretation of analysis results. Once entered, VPG's full complement of post processing functions allow the user to graphically display and manipulate solver code result files with contour and deformation animation, contour plots and fills, and geometry deformation.

Continued on next page.

VPG's post processing functions allow the user numerous ways of animating and viewing the analysis results. The user has the options of animating select frames, viewing single frames, altering the time step, rotating the model, or viewing only select parts, of the model.

To utilize the post processing functions, users generate a result file with the suffix .pp (post processing). The .pp file is generated by reading a result file into the post processor. The post processing function automatically prompts the user for the result file when activated. VPG post processes the following types of result data:

D3PLOT (d3plot01, etc.)
NASTRAN PUNCH (.pch)
DYNA DEFORMED GEOMETRY (.defgeo)
NASTRAN PACKED PUNCH (.pac)
HISTORY (.his)

Once the .pp file is created it is recommended that the user save it. It requires less space than the analysis result file used to create it and can be used for all subsequent post processing functions. It also requires significantly less time to load the .pp file. Once, the .pp file is created, the file used to generate it can be deleted (in most cases).

FATIGUE

VPG allows the user to generate and post process fatigue result files from within the VPG interface.

ANIMATE CONTOUR

This function is used to map the stresses, strains and strain energy of the model across time. Animations can be edited to even, odd, or specified frames.

ANIMATE DEFORMATION

This command allows the user to animate displacements within the model in real time. Animations can be edited to even, odd, or specified frames.

CONTOUR FILL

This function maps stresses, strains, and strain energy in the model for a single step. It allows the user to view the contour values by superimposing a fill-color contour image onto the model.
The values are displayed in a color legend in the upper right hand corner of the screen.

Continued on next page.

CONTOUR LINES

This function, similar to the CONTOUR FILL function, allows the user to check the model's contour values for a single step. A color, wire frame, contour-line plot of the component result is superimposed onto the current model.

DEFORMED SHAPE

This command displays the displacement results of the model for a single step. The undeformed model shape is displayed in white. The model can be animated to show the transition between the undeformed and deformed model.

ELEMENT STRESS

This function displays the stress results of each individual element as opposed to the contour plot functions which display the stress results in terms of the average stress at each node. A color legend for the corresponding color values is displayed in the upper right hand corner of the screen.

TIME HISTORY PLOT

eta/VPG's TIME HISTORY PLOT functions enable the user to visualize the results of an analysis with XY plots. VPG offers a wide range of tools to manipulate the information on the display screen with labels, colors, multiple graphs, and a host of advanced filtering techniques e.g., FIR, Butterworth, SAE, scaling, smoothing, and averaging.

FFT (FAST FOURIER TRANSFORM

The TIME <-->FREQUENCY function allows the user to convert time domain to frequency domain response for signal processing analysis. The FFT equation is used to convert time domain to frequency domain and frequency domain to time domain for any graph plot.

2.3 MENU SYSTEM

The program starts in the **MAIN** menu (see figure on page 20) and branches out into sub-menus. The user selects a sub-menu by mouse pick or keyboard entry. Descriptions for these menu options are located in their respective sections.

FILE	Imports and exports data to and from VPG.

Continued on next page.

PREPROCESSOR	Contains a menu of preprocessing functions.
LINE	Creates and modifies line/surface data.
SURFACE OPTIONS	Creates and modifies surfaces in VPG.
ELEMENT OPTIONS	Creates and modifies elements.
NODE OPTIONS	Creates, copies, transforms, and manipulates nodes.
MODEL CHECKER	Checks element criteria (warpage, boundary, aspect ratio, etc.).
SET MENU	Creates node and element sets for superelement files and substructure files, etc.
BOUNDARY CONDITIONS	Creates and verifies constraints and loads on a finite element model.
MATERIAL PROPERTY	Creates and assigns material properties.
ELEMENT PROPERTY	Creates and assigns element properties.
CONTACT INTERFACE	Creates and modifies sliding/rigidwall interfaces for LS-DYNA.
DYNA MISCELLANEOUS	Handles distinct LS-DYNA miscellaneous data.
SUPER ELEMENT	Assigns and modifies SUPERELEMENT ID numbers (only for NASTRAN application).
ROAD MENU	Defines road surfaces.
SUSPENSION MENU	Defines and auto-generates front and rear automotive suspensions.
TIRE MODEL	Defines and auto-generates tire models.
ANALYSIS	Analyzes proving ground events.
FATIGUE	Analyzes element fatigue.
FFT	Provides signal processing.
POST PROCESSING	Provides options for viewing the results of an analysis.
GRAPH	Plots dynamic characteristics of the structure vs. time, velocity, etc.
RESTART	Restarts a current VPG session with a current or another database.
SESSION FILE	Regains lost work (e.g., after a system

Continued on next page.

	crash) lost during a VPG session.
VIEWING OPTIONS	Manipulates the display, position, and perspective of a model.
EXIT PROGRAM	Exits VPG.

2.4 FUNCTION KEYS

Function keys 1 through 8 direct the user to the most frequently used menus. The F1 (function key 1) is reserved for the Main Menu.

F1	Main Menu	F5	Model Checker
F2	Element Option	F6	Node Options
F3	File	F7	Surface Options
F4	Line	F8	Free Rotation

2.5 DISPLAY WINDOW

VPG breaks the screen into five distinct regions. The regions are used to receive input or display messages for the user. The five regions are illustrated on the following page.

1. DRAWING WINDOW -- Models and definition cards are displayed in this area.
2. VIEWING OPTIONS WINDOW-- This group of commands dynamically manipulate the display, position, and perspective of a model.
3. MENU WINDOW -- Commands and command options are displayed in this area. They can be accessed via the keyboard mouse.
4. DIALOGUE WINDOW -- VPG displays comments and messages to the user and accepts keyboard entry commands in the dialogue window.
5. DISPLAY PARAMETER OPTIONS WINDOW -- These commands set the plot options for the current model.

Continued on next page.

2.6 MOUSE FUNCTIONS

All VPG functions are accessible via the left mouse button. To access a function the user selects the desired button using the mouse pointer and depresses the left mouse button. This button is also used for selecting definition cards, locating the cursor in definition cards, creating drag windows, locating points, nodes, elements, etc. The right mouse button activates a pull-down menu with the following commonly used functions:

FREE ROTATION
ZOOM
PAN
FILL
CLEAR
ACTIVE WINDOW ON/OFF

Continued on next page.

VIRTUAL - X ROTATION
VIRTUAL - Y ROTATION
VIRTUAL - Z ROTATION
SCREEN ROTATIONS
FIXED VIEWS
NODE OPTIONS
ELEMENT OPTIONS
GEOMETRY BUILDER

2.7 KEYBOARD ENTRY

To increase speed and efficiency all VPG functions can also be accessed by keystroke entry. Entering a one or two-letter combination followed by the return key activates each command of the menu that the user presently has on the screen. For main menus, the letter combination is the first two letters of a one word command or the first letter of each of the first two words of a two or more word command. For example, the keystroke entry for the command FREE ROTATION in the above menu would be “f”, “r” followed by the RETURN key. As the user types the keys, the matching command will be highlighted. For the ZOOM command, the keystroke entry would be “z”, “o” followed by the RETURN key. For control keys, the user need only type the first letter to access the function.

2.8 SPECIFICATIONS

The standard version of VPG has the following specifications for UNIX based workstations per database (VPG-Large is available to handle extra-large models):

150,000	LINES
200,000	POINTS
4,000	SURFACES
	-200,000 Edge Points (SURFACE)
	-400,000 Control Points (SURFACE)
125,000	GRIDS
125,000	ELEMENTS
1,000	PROPERTIES
1,000	PIDS

2.9 LINE DATA

VPG's built-in translator converts and filters line data from the following programs into a neutral line format:

IGES (line and surface)
CGS (DES, INCA)
DXF file formats and
PDGS (standard format)
VDA file

2.10 CONVENTIONS

This manual is designed to reduce the amount of reading material on the page and maintain text clarity. Several fonts and symbols are implemented throughout the manual. An example is given at the bottom of the page.

FIXED FONT (ALL CAPS, BOLD)	This font indicates text found within VPG e.g., menu names, subsections, commands, and options within commands, etc.
Proportional Font	This font indicates explanatory text e.g., command descriptions, notes, and section titles.
ALL CAPITALS	This font indicates a function, menu name, card, command, etc. found in explanatory text.
>	The greater than symbol directs the user to read the text displayed in VPG's DIALOGUE WINDOW.
?	The question mark directs the user to select an option that is listed in VPG's MENU WINDOW.
•	The bullet signals a description of the previous command or situation.

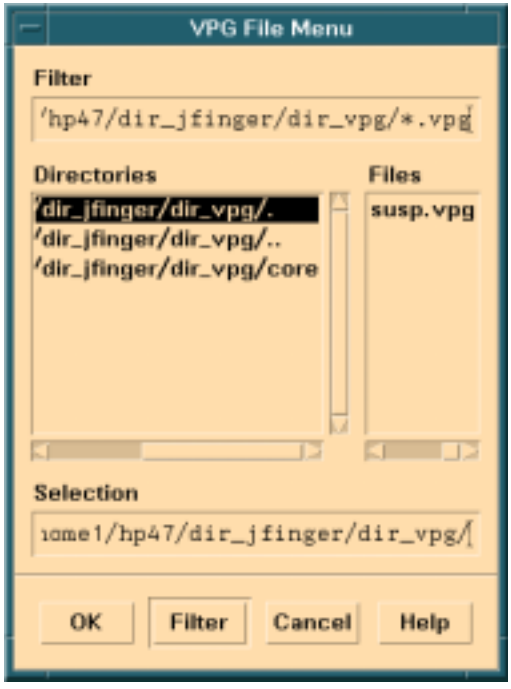
-CREATE
4 - PLATE ELEMENT

This section covers the options for the PLATE ELEMENT subsection of CREATE ELEMENTS.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   ? NODE
   POINT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three.
   - An element will be created after three or four nodes/points are selected.
   - EXIT or ABORT will exit this function.

2.11 VPG FILE MENU

The VPG FILE MENU allows users to easily access files and directories. The DIRECTORIES and FILES windows are accompanied by scrollbars and the VPG FILE MENU window is expandable for easy viewing of directories and files.



All files listed in the FILES window are displayed from the current directory based on the filter (filename extension such as .**vpg**). The current directory is listed at the bottom of the DIRECTORIES window. The filter is set by changing the filter parameter in the FILTER window and clicking the filter button. The SELECTION window allows the user to enter the file name and directory manually.

2.12 RECOMMENDED NAMING CONVENTION (.his, .lin, .bin, etc.)

The protocol for naming files during a VPG session includes attaching suffixes to the file names which specify the file types.

Examples of suffixes include:

1. VPG database file name: *filename*.**vpg**

2.	NASTRAN file name:	*filename*.**dat**
3.	VPG line data file name:	*filename*.**lin**
4.	VPG binary database file name:	*filename*.**bin**
5.	VPG binary result file name:	*filename*.**pp**
6.	LS-DYNA database:	*filename*.**dyn**
7.	LS-DYNA history result file:	*filename*.**his**

Example: When reading in a line data file, VPG prompts for a line data file name (all file names in that directory with the suffix **.lin** are listed in the options area). The user then selects the appropriate file name. This practice makes the file name selection conve nient and organizes the user's work directory.

2.13 LS-DYNA CARDS

eta/VPG supports all LS-DYNA cards. As a result, card definition requires no text editing. The cards are arranged as they are in the LS-DYNA manual. eta/VPG displays the cards in the conveniently edited window seen on the opposite page and described below.

CARD SELECT BUTTON

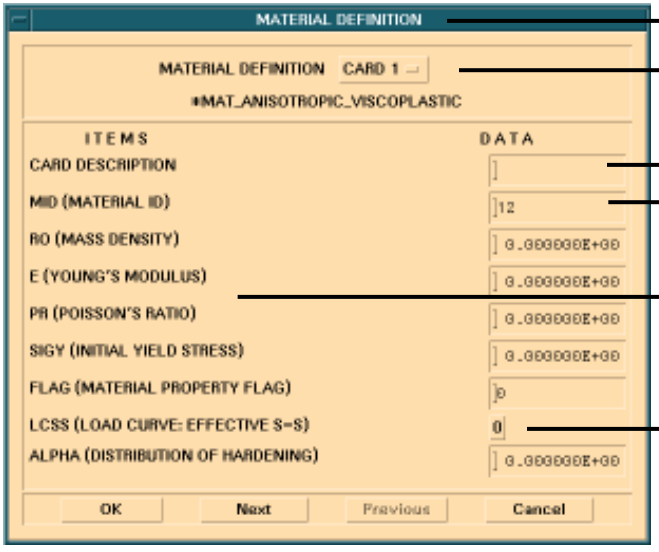
The CARD SELECT BUTTON displays the number of definition cards and allows the user to jump conveniently to the desired card. The card button is not an option when only one card is needed to define the selected property or the number of cards is dependent on user defined values.

CARD DESCRIPTION FIELD

The CARD DESCRIPTION FIELD allows the user to enter a name for the defined properties.

EDITING FIELD

VPG's smart editing field only allows values within the specified range to be entered. If the value entered is beyond the LS-DYNA defined range VPG prompts the user when the user tries to move to another field or card.



FIELD DESCRIPTION

The field description indicates the value to be added to the editing field. If the field description text is gray, it means that the value entered in that field is dependent on another field which must be edited first.

EDITING PUSH BUTTON

An editing button forwards the user through a series of prompts, menus, or cards which results in a valid value for the chosen field.

OK

Accepts and saves the defined card.

NEXT

Forwards the user to the next card.

PREVIOUS

Forwards the user to the previous card.

CANCEL

Exits without saving or defining the card.

ENTER

Pressing ENTER at any time during card definition accepts and saves the defined card.

2.14 LOCAL COORDINATE SYSTEM

VPG refers to the local coordinate system to translate, rotate, mirror, copy, and generate points, lines, or nodes. When such a function is selected, the program will automatically prompt the user to generate a local system designated as the UVW coordinate frame.

Either one, two, or three reference points are required to establish a local coordinate system. For a three point system, the first reference point defines the local origin. The second reference, which extends from the first reference point, defines the direction of the local U-axis. The third reference point defines the local UV plane. The local W-axis is defined in the UV plane and is perpendicular to the U-axis. The local W-axis is then defined according to the right hand rule perpendicular to the UV plane.

For the two reference point option, DONE should be selected after the second reference point is defined. The local W-axis lies along the vector from the origin to the second reference point; the V-axis lies in the VW plane; and the U-axis is defined by the right hand rule.

note: All rotational commands (generating arcs, copying with rotation, etc.) are executed about the local W- or global Z-axis.

For a one reference point option, the user selects a point or node on the screen as the local origin, then enters one of the X, Y, or Z options that are listed to define the local W along one of the global axes.

note: Defining a local coordinate system with "one" reference point is described in the following text and marked with "•."

1. When creating a local coordinate system, VPG prompts:

   > CREATE LOCAL COORDINATES
   PICK NODE/PT FOR ORIGIN

   • VPG prompts for a desired coordinate system (local or global) in the options area.

   ? ABORT
   GLOBAL SYSTEM
   KEY IN X, Y, Z

   • This signals the user to key in the origin of the local coordinate system.

   > ENTER X, Y, Z COORDINATES

- The user enters the values for the X, Y, and Z coordinates, e.g., 100,0, and 0.

NODE

- Default

POINT

SHOW LAST C.S.

- Once the last coordinate system has been displayed, see step 3.
- The user may select a node or a point, enter a coordinate by key board, or use any combination of these options to create a local coordinate system.

2. Once a reference point has been selected, VPG prompts:

> PICK NEXT POINT OR NODE

? ABORT

DONE

- After the user selects 2 reference points, VPG displays the coordinate system and continues to step 4.

KEY IN XYZ

- The user may enter up to 3 reference points globally to define a local coordinate system.

INCREMENTAL XYZ

- The user may enter DX, DY, and DZ from the previous reference point to define a local coordinate system.

NODE

- The user may select up to 3 nodes to define a coordinate system.

POINT

- The user may select up to 3 points to define a coordinate system.

REJECT LAST

- This allows the user to deselect the last reference point during the selection process.

X AS LOCAL W AXIS

- This defines a local coordinate system that is parallel to the global axis with the local W along the global X-axis and the local origin at the first reference point.

Y AS LOCAL W AXIS

- This defines a local coordinate system that is parallel to the global axis with the local W along the global Y-axis and the local origin at the first reference point.

Z AS LOCAL W AXIS

- This defines a local coordinate system parallel to the global axis with the local W along the global Z-axis and the local origin at the first reference point.

3. Once the user defines the desired coordinate system, VPG displays it and prompts:

   > ACCEPT ? (Y/N/A)

   ? YES

   - VPG prompts for the next command.

   NO

   - The user returns to step 1.

   ABORT

   - The user returns to the menu.

2.15 ENTITY SELECTION

In certain commands such as COPY, DELETE, etc., VPG prompts the user to select elements, nodes, lines, surfaces, etc. A list of options will appear in the MENU WINDOW. The default selection option is cursor pick at the entity. Other commonly used selection options are described below:

WINDOW

A window (drag-window) is defined by clicking the left mouse button, dragging the cursor diagonally across the screen until the desired entities are within the window and then clicking the left mouse button a second time to complete the selection. If an entity is partially outside the window, it will not be selected.

MULTI-PT REGION

A multi-point region (polygon) is defined by clicking the left mouse button in succession to enclose the desired polygon region. Click on the right mouse button to reject the last defined point. Click on the middle button to complete the region. Entities within this region will be selected. If part of an entity is outside of the region, it will not be selected.

PART

The part names will be listed in the menu area. Pick the name from the part list or pick an entity from the screen to select the part. Selected parts will be highlighted in white in the MENU WINDOW. All entities in the part will be selected.

REJECT

This option negates the last selection whether from single cursor pick or a group of entities selected by any of the above options.

EXCLUDE ON/OFF

This option works like a toggle switch. If turned on, all the subsequent selected entities will be removed from the previously selected list. The user may toggle this option on and off during the course of the selection.

TYPE

This option is used to control the type of elements to be selected. If DONE is selected immediately after selecting TYPE, all elements of this type will be selected. Otherwise, other options may be used to limit the selection of elements.

note: The TYPE function is specific to the selection of elements.

SURFACE

This function is used to select elements created from a surface by the SURFACE MESH command.

note: The SURFACE function is specific to the selection of elements.

3.1 OPENING/CREATING an eta/VPG DATABASE FILE

Type **'vpg'** in a Unix window to start the program. Once the windows are activated, the VPG FILE MENU window is displayed for the user to OPEN or CREATE a new VPG database.

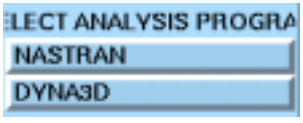
1.

2. The user would either select the name of a previously saved file or enter the name of a new file in the Dialogue window. The recommended practice is to add the extension **.vpg** to a newly created file.

Continued on the following page.

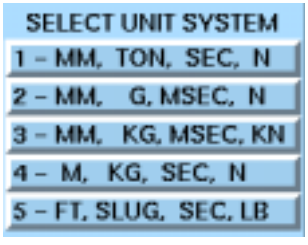
3. If creating a NEW file, the user would be prompted to do so:

4. The user will be prompted to select the analysis program desired:



The analysis program selected will set defaults for the eta/VPG session to generate either LS/DYNA or NASTRAN cards.

5. The user will also be prompted to select the Unit System desired:



The selected unit system will be stored in the database as the default value setting. The user will now be in the MAIN MENU of eta/VPG and ready to start the session.

Continued on next page.

3.2 SETTING UP A VPG MODEL

Following is a general outline for setting up a VPG model. There are three unique modules in the VPG pre-processor that allow the user to create a VPG model in an automated procedure. A detailed description of each of these processes is available in the VPG training manual.

1- Auto-Generating a Front Suspension Model

The user begins by selecting a front suspension model from the SUSPENSION MENU. The user determines the desired optional components such as stabilizers and steering system types and enters the node coordinates for suspension geometry. Next the user will be prompted to edit the default spring stiffness, damping coefficients, extra node coordinates, default mass, center of gravity and inertia moment. Once the user has edited the default suspension values, the suspension will be displayed on screen.

2- Auto-Generating a Rear Suspension Model

The rear suspension is generated in the same way that the front suspension is.

3- Auto-Generating Tire Models

After entering the TIRE MENU the user selects GENERATE and is offered the TIRE PARAMETER dialogue box. Default values for tire geometry, mass and inflation pressure are edited to the user's parameters. Once the tire geometry is defined, VPG prompts the user for the location of the tire. The spindle of one of the suspensions is selected and the tire is attached. The user then selects the other three spindles in clockwise or counter-clockwise fashion and the remaining three tires are generated and added to the model. Once the tires are defined, the user attaches them to the suspension, defines airbags for tire pressure and defines the initial rotational velocity of the tires.

4- Auto-Generating a Road Surface

The user generates the desired road surface by selecting it from the ROAD SURFACE MENU. If the road surface is not in the correct position in relation to the suspension after generation, the MOVE ROAD SURFACE command is used to position it. Next, contact between the road surface and the tires is determined using the FIVE_NODES_TO_SURFACE interface type. The road is then constrained in the Y, Z and rotational directions using material property assigned to the road. Next, a velocity is assigned to the road using the BOUNDARY_PRESCRIBED_MOTION card.

5- Reading the Simplified Body Model

The body to chassis/suspension attachment process depends upon what type of body model the user wishes to use for analysis (deformable or rigid).

Rigid Body- The rigid body model is constrained to the rigid beams that define the body attachment points on the suspension.

Deformable Body- The specific coordinates for the body attachments points must be entered when the user defines the Extra Node Coordinates for the front/rear suspension models. This ensures that the generated suspension would fit the specific body model. The user then creates weld spiders between the mounts on the vehicle and the rigid body beams on the suspension.

Next the user defines the BODY_LOAD_DEFINITION_CARD to define the gravity.

After defining the gravity the user pre-loads the suspension using the ELEMENT DEFINITION CARD.

6- Defining VPG Analysis Control Parameters

First, the CONTROL ENERGY and the CONTROL TERMINATION cards must be defined. Next the output control interval data for the ASCII database must be determined. Then the BINARY DATABASE cards must be edited to control the output interval of the results and restarts.

7- Analysis Submission

From the ANALYSIS menu, select DYNA INPUT FILE OPTIONS, edit the dialogue box, and submit the analysis.

8- Displaying Results

When post-processing the analysis results, RESTART the eta/VPG session, and CREATE A NEW FILE called ‘pp.vpg’- pp for post-processing.

Enter the POST PROCESSING menu and select D3PLOT (LS/DYNA result file). Then select the analysis' ‘d3plot’ from the File Menu. Select LS/DYNA version 940. Select ALL AVAILABLE STEPS. Select ALL COMPONENTS

The results from each of the analysis steps will then be read into eta/VPG. A binary result file will be created at this time (named ‘d3plot.pp’). Since the results are not saved to the eta/VPG database, this binary file should be re-read into the post-processing menu when the user wishes to view the results again. This file is read much faster than the d3plot files.

The results are now ready to be post processed using a variety of features.

9- Graph Plotting

To graph the results, select GRAPH from the Main Menu. The user then has the option of reading in a previously saved LS-DYNA ASCII graph file. When exiting the GRAPH MENU, the user has an option to save the graphs in a binary file for future processing. This file will be loaded mush faster than the LS-DYNA ASCII files.

Continued on next page.

MAIN MENU

This is the first menu the user encounters when beginning a session with VPG. The initial options unfold into an additional series of submenus. These submenus are documented in the following sections.

MAIN MENU

FILE
PRE-PROCESSOR
ROAD
SUSPENSION
TIRE MODEL
ANALYSIS
FATIGUE
POST PROCESSING
GRAPH
RESTART
SESSION
VIEW
EXIT PROGRAM

FILE

The options available in FILE MENU allow the user to input data from different analysis programs into VPG.

FILE MANAGER

READ ABAQUS FILE
READ BINARY FILE
READ C-MOLD FILE
READ LS-DYNA FILE
READ F-CRASH FILE
READ GENESIS FILE
READ LINE DATA
READ MOLDFLOW FILE
READ NASTRAN BULK
READ PAMCRASH FILE
READ RADIOSS FILE
READ SUSP(ADAMS)

WRITE ABAQUS FILE
WRITE BINARY FILE
WRITE C-MOLD FILE
WRITE LS-DYNA FILE
WRITE F-CRASH FILE
WRITE GENESIS FILE
WRITE LINE DATA
WRITE MOLDFLOW FILE
WRITE NASTRAN BULK
WRITE PAMCRASH FILE
WRITE RADIOSS FILE
SUMMARY REPORT
VIEW
EXIT

A detailed description of each function is given in the following section.

READ ABAQUS FILE

VPG reads ABAQUS files directly with its built in translator. The VPG ABAQUS file extension is **.inp**.

READ BINARY FILE

This function allows the user to read a VPG binary file (**.bin**) into VPG. The binary model file contains only the finite element model data written by VPG. Any number of binary files may be read in at any time during the VPG session. VPG allows the user to offset the node/element numbers if the numbers have already been used in the database.

1. VPG prompts:
   > READING VPG MODEL BINARY INPUT FILE
   > ENTER FILE NAME
   - The files with the suffix **.bin** will be listed in the VPG FILE MENU.

2. If the file does not exist the prompt reads:
   > FILE DOES NOT EXIST
   > RE-ENTER OR STOP

3. If the user is reading data into an existing model, VPG prompts:
   > OFFSET NODE AND ELEMENT NUMBERS? (Y/N)
   - YES renumbers the new data at the lowest available unused node/element number.
   - NO compares the node/element numbers and ignores the duplicates--VPG retains only the original nodes and elements (i.e., if the binary file contains all duplicate node and element numbers, the new part will read in without the presence of elements or nodes).

4. VPG reads in the file and returns the user to the FILE MANAGER menu.

READ C-MOLD FILE

This function allows the user to read a C-MOLD (.**fem**) file directly into VPG.

1. VPG prompts:
   > READING C-MOLD INPUT FILE
   - The files with the suffix **.fem** will be listed in the options area.

2. After the user enters a filename, VPG prompts:
   > SI UNIT IS USED IN FEM FILE
   > SELECT UNIT TO BE USED IN VPG DATABASE
   ? MM
   CM
   METER
   IN
   EXIT
   - This sets the measurement unit for the model.

3. If the user is reading data into an existing model, VPG prompts:
   > OFFSET NODE AND ELEMENT NUMBERS? (Y/N)
   - **YES** renumbers the new data at the lowest available unused node/element number.
   - **NO** compares the node/element numbers and ignores the duplicates--VPG retains only the original nodes and elements (i.e., if the c-mold file has all duplicate node and element numbers, the new part will read in without the presence of elements or nodes).

4. VPG reads in the file and returns the user to the **FILE MANAGER** menu.

READ LS-DYNA FILE

This function allows the user to read both keyword and non-keyword LS-DYNA (.**dyn**) files directly into VPG. VPG supports versions 88 to 940 Keyword. Once the user has entered the command, the files with the suffix **.dyn** will be listed in theVPG FILE MENU. The user can then select the desired file.

READ F-CRASH FILE

This function allows users to read F-CRASH (**.fcr**) files directly into VPG. Once the user has entered the command, the files with the suffix **`.fcr`** will be listed in the options area. The user can then select the desired file.

READ GENESIS FILE

This function allows users to read GENESIS (**.dat**) files directly into VPG. Once the user has entered the command, the files with the suffix **`.dat`** will be listed in the options area. The user can then select the desired file.

READ LINE DATA FILE

This function allows users to read converted wireframe and IGES surface data into VPG. In order to provide flexibility in accepting line data from different CAD systems, VPG uses a neutral line data format to communicate with these CAD systems: IGES, PDGS standard, CGS (INCA and DES), and DXF. Once the user has entered the command, the files with the suffix **`.lin`** will be listed in the options area. The user can then select the desired file.

note: VPG MENU provides the necessary translators for the file types listed above. The filter programs provided in the VPG MENU reduce the number of unnecessary points in the CAD line data, combine duplicate lines, and join line segments into continuous lines.

READ MOLDFLOW FILE

This function allows users to read MOLDFLOW data directly into VPG.

1. VPG prompts:
   `> READING MOLDFLOW INPUT FILE`
   `> ENTER THE ROOT NAME OF THE MOLDFLOW INPUT FILES`

2. After entering the root file name (assuming that the **`.mfl`**, **`.mod,`** and **`.tri`** files are available) VPG displays the MOLDFLOW model.

3. If the user is reading data into an existing model, VPG prompts:

Continued on next page.

> OFFSET NODE AND ELEMENT NUMBERS? (Y/N)

- **YES** renumbers the new data at the lowest available unused node/element number.
- **NO** compares the node/element numbers and ignores the duplicates--VPG retains only the original nodes and elements (i.e., if the MOLDFLOW file contains all duplicate node and element numbers, the new part will read in without the presence of elements or nodes).

4. VPG reads in the file and returns the user to the **FILE MANAGER** menu.

READ NASTRAN BULK

VPG uses NASTRAN as a file translator so that the user may import and export models and mesh. This function allows the user to read a NASTRAN bulk data file (**.dat**) directly into VPG. All existing properties, materials, and subcases are retained.

1. VPG displays the VPG FILE MENU and prompts:
   > DEFINE NASTRAN BULK DATA FILE

2. VPG prompts:
   > SELECT PART CONTROL OPTION FOR CBARS
   - The files with the suffix **.nas** will be listed for you in the options area.

   ? MAT1 ID
   - CBARS will be grouped by common MAT1 ID.

   PBAR ID
   - CBARS will be grouped by common PBAR ID.

   PART NAME
   - CBARS will be grouped by their individual part names.

3. If the user is reading data into an existing model, VPG prompts:
   > OFFSET NODE AND ELEMENT NUMBERS? (Y/N)
   - YES renumbers the new data at the lowest available unused node/element number.
   - NO compares the node/element numbers and ignores the duplicates--VPG retains the only original nodes and elements (i.e., if the NASTRAN file contains all duplicate node and element numbers, the new part will read in without the presence of elements or nodes).

Continued on next page.

4. VPG reads in the file and returns the user to the FILE MANAGER menu.

READ PAMCRASH FILE

This function allows the user to read a PAMCRASH (.**pc**) file directly into VPG. Once the user has entered the command, the files with the suffix **.pc** will be listed in the options area. The user can then select the desired file.

READ RADIOSS FILE

This function allows the user to read a RADIOSS data file directly into VPG. VPG can read fixed format input files from RADIOSS Versions 2.1, 2.2, 2.3, 3.1, and 4.1 VPG can write RADIOSS 4.1 fixed format input files. VPG supports all input cards of RADIOSS 4.1 fixed format. VPG contains options when reading RADIOSS input and output files. These options relate to the translation of rigid bodies to VPG and the loading of model information. RADIOSS requires two files to be loaded- one is the RADIOSS output file (.out) which contains the rigid body information. The second is the RADIOSS input file (.D00) which contains model information. If the user has both files, he should click YES at the first prompt to read both the RADIOSS output and input files. If the user has only the RADIOSS input file, the user should select NO.

1. Begin with reading the RADIOSS output file (.out). This loads the rigid body information. VPG will prompt:
   > ADJUST RIGID BODY PRIMARY NODES FROM RADIOSS OUTPUT? (Y/N)
   - If **YES** is selected, the RIGID BODY information will be loaded with the output file. The user can then load the RADIOSS input file (.D00).
   - If **NO** is selected, VPG will prompt:

   > ADJUST RIGID BODY PRIMARY NODES BY VPG? (Y/N)
   - **YES** will locate the RIGID bodies and recalculate the rigid body primary node locations.
   - **NO** will finish the input sequence without modifying the model information.

VPG creates some default materials and element properties for elements that are lost during the file reading. These are assigned to the part, DEFAU_#. VPG creates a node set, element set, material set, property set, and interface (contact) set for boundary conditions, interface, rigid walls, time history etc. VPG can read multiple RADIOSS input files into the same database.

READ SUSP (ADAMS)

Section J of the Appendix gives a complete description of converting ADAMS models to VPG. Please refer to that section when converting an ADAMS suspension.

WRITE ABAQUS FILE

VPG displays the CONTROL PARAMETER PopUp window. Users may select the parameters with the mouse button and enter new values through the keyboard. (The parameters are ABAQUS specific. Refer to the ABAQUS manual for more details.)

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters). Using the **.inp** convention for ABAQUS files is suggested.

2. If the file name already exists, VPG prompts the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. If materials and properties have already been defined, a "**CONTROL PARAMETERS**" window will be shown and VPG prompts:
   > DEFINE/MODIFY DATA
   > SELECT AN ITEM TO EDIT
   - EXIT **sends user to step 6.**

4. If there are no materials or properties defined in the database, VPG prompts:
   > NO MATERIALS DEFINED IN DATABASE! SELECT OPTION
   ? 1 - SELECT GLOBAL DEFAULT
   - Defines one global material and one global property for the database.

   2 - DEFINE BY PARTS
   - Assigns materials and properties to each individual part.

   3 - IGNORE DEFINITION
   - Ignores material and property definition.

   4 - EXIT

5. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   - **YES** will only output the active parts (parts that are turned ON).
   - **NO** will output all parts in the database.

6. VPG prompts:
   > ENTER TITLE
   - The user can enter up to 60 characters.

WRITE BINARY FILE

This function allows the user to output a binary model file from current model data. The binary model file may be used to combine finite element models from different VPG data files. This function will output elements and nodes only.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters). Using the **.bin** convention for BINARY files is suggested.

2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - YES will overwrite the existing file.
   - NO will prompt for a file name.

3. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   - YES will only output the elements displayed on the screen.
   - NO will output all elements existing in the database.

4. VPG returns the user to the FILE MANAGER menu.

WRITE C-MOLD FILE

This function allows the user to output a C-MOLD (**.fem**) file from current model data.

1. VPG prompts:
   > WRITING C-MOLD INPUT FILE
   SELECT UNIT CURRENTLY USED IN DATABASE
   ? MM
   CM
   METER
   IN
   EXIT

2. VPG echoes the user's unit selection and prompts for the new file's name:
   > SI UNIT WILL BE USED IN THE FEM FILE, SCALE FACTOR = X.XXXXXX
   > ENTER FILE NAME OR "STOP" TO EXIT

3. If there are no materials or properties defined in the database, VPG prompts:
   > NO PROPERTIES DEFINED IN DATABASE! SELECT OPTION
   ? 1 - SELECT GLOBAL DEFAULT
   - Defines one global material and one global property for the database.

   2 - DEFINE BY PARTS
   - Assigns materials and properties to each individual part.

   3 - IGNORE DEFINITION
   - Ignores material and property definition.

   4 - EXIT

4. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   - **YES** will only output the active parts (parts that are turned ON).
   - **NO** will output all parts in the database.

5. VPG prompts:
   > ENTER TITLE
   - The user can enter up to 60 characters.

Continued on next page.

6. If there are no model properties defined in the database, VPG prompts:
   > NO NODAL PROPERTIES DEFINED IN DATABASE
   > ENTER NEW PROPERTY NAME OR "EX" TO EXIT
   - A "**NODAL PROPERTY**" window appears after a name is selected, and VPG prompts the user to modify data in the window.
   - **EXIT** ends this function and returns the user to the **FILE MANAGER** menu.

WRITE LS-DYNA FILE

This function allows the user to output a LS-DYNA data deck directly from the VPG database.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters). The **.dyn** extension is suggested for LS-DYNA files.

2. If the file name already exists, the program will prompt the following message:
   > FILE NAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - YES will overwrite the existing file.
   - NO will prompt for a file name.

3. VPG displays the current unit system and prompts:
   > SELECT UNIT SYSTEM FOR OUTGOING DATA
   1 - MM, TON, SEC, N
   2 - MM, G, MSEC, N
   3 - MM, KG, MSEC, KN
   4 - M, KG, SEC, N
   5 - ORIGINAL UNIT

4. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   - **YES** will only output the elements displayed on the screen.
   - **NO** will output all elements existing in the database.

Continued on next page.

5. VPG prompts:
   > ENTER TITLE

6. After entering the title, VPG displays the CONTROL TERMINATION CARD if a termination time has not been specified yet. Once the card is defined and the user selects OK the DYNA deck is written.

note: If the properties/materials are not assigned to the parts in the database, VPG prompts the user to define the properties/materials by default, by global default, or by part.

WRITE F-CRASH FILE

This function allows the user to output a F-CRASH (.fcr) data deck directly from the VPG database.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters).

2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. VPG prompts:
   > ENTER TITLE

4. Parameter windows appear and VPG prompts:
   > DEFINE/MODIFY DATA
   > SELECT AN ITEM TO EDIT

Continued on next page.

5. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   ? YES
   - Outputs parts currently turned on.

   NO
   - Outputs entire database.

note: If the properties/materials are not assigned to the parts in the database, VPG prompts the user to define the properties/materials by default, by global default, or by part. Supported properties are listed in Appendix D, *VPG Capabilities for F.E.A. Programs.*

WRITE GENESIS FILE

This function allows the user to output a GENESIS (.dat) data deck directly from the VPG database.

WRITE LINE DATA FILE

This function allows the user to output a LINE/SURFACE DATA (**.lin**) data file directly from the VPG database.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters). The **.lin** extension is suggested.

2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   ? YES
   NO
   - YES will overwrite the existing file.
   - NO will prompt for a file name.

Continued on next page.

3. VPG prompts:
   > WRITE LINES/SURFACES ONLY IN ACTIVE PARTS? (Y/N)
   ? YES
   NO
   - YES will only output the lines/surfaces displayed on the screen.
   - NO will output all elements lines/surfaces in the database.

WRITE MOLDFLOW FILE

This function allows the user to output the current model data into an external file in MOLDFLOW format.

note: Node and element numbers must be in sequence to write a MOLDFLOW file. The user may either renumber or compress nodes/elements (see **NODE OPTIONS** menu). If the nodes are not in sequential order, VPG prompts:
> RENUMBER NODE NUMBERS AND TRY AGAIN (or)
> RENUMBER ELEMENT NUMBERS AND TRY AGAIN

1. VPG prompts:
   > ENTER ROOT NAME FOR INPUT FILES OR STOP TO EXIT
   - VPG writes three files in accordance with the MOLDFLOW format (**.mfl**, **.nod,** and **.tri**).

2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. After entering file name, VPG prompts:
   > SELECT UNIT CURRENTLY USED IN DATABASE
   ? MM
   CM
   IN

Continued on next page.

4. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   - **YES** will only output the elements displayed on the screen.
   - **NO** will output all elements existing in the database.

5. VPG prompts:
   > ENTER TITLE

WRITE NASTRAN

This function allows the user to output the current model data to an external file into NASTRAN format. The user may output either the complete model data or partial model data.

1. VPG prompts:
   > ENTER CONTROLS OPTION
   ANALYSIS TYPE
   - Displays a submenu with selective analysis types:
     STATICS
     NORMAL MODES
     BUCKLING
     FREQUENCY RESPONSE
     TRANSIENT RESPONSE

   FORMAT OPTION
   - Displays a submenu with file format options:
     SINGLE FIELD (fixed format columns of eight)
     DOUBLE FIELD (double precision)
     FREE FORMAT

   OUTPUT OPTION
   - Displays a submenu with ouput options:
     PRINT
     PUNCH
     ALL

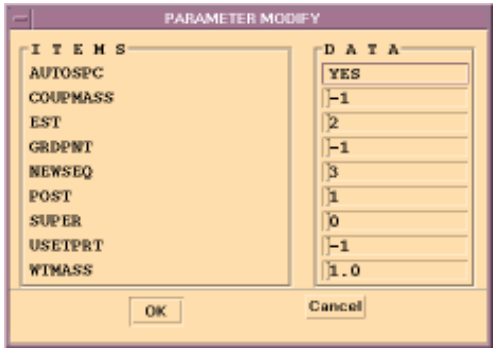
Continued on next page.

PARAM CARDS

- Displays the parameter modify window (see figure below).

WRITE FILE

EXIT



2. VPG prompts:

   > ENTER FILE NAME OR "STOP" TO EXIT

   - Enter a file name (up to 24 characters).

3. VPG prompts:

   > OUTPUT ONLY ACTIVE PARTS? (Y/N)

   ? YES

   NO

   - YES will only output the elements displayed on the screen.
   - NO will output all elements existing in the database.

4. VPG prompts:

   > ENTER ANALYSIS TITLE

   - Any alphanumeric input up to 72 characters may be entered as the title of the NASTRAN bulk data file.

WRITE PAMCRASH FILE

This function allows the user to output a PAMCRASH (.**pc**) data file directly from the VPG database.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters).
2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. VPG prompts:
   > SELECT PAMCRASH VERSION:
   ? 1 - SEQUENTIAL
   2 - FREE NUMBERING
   > DO YOU WANT TO USE KEYWORD INPUT(Y/N)
   ? YES
   NO

4. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   ? YES
   NO

5. VPG prompts for optional keyword cards via pop ups.

6. VPG prompts:
   > ENTER TITLE
   - The user enters an alpha-numeric title containing a maximum of 48 characters.
   - VPG returns the user to the FILE MANAGER menu.

note: If the properties/materials are not assigned to the parts in the database, VPG prompts the user to define the properties/materials by default, by global default, or by part. Supported properties are listed in Appendix D, *VPG Capabilities for F.E.A. Programs.*

WRITE RADIOSS FILE

This function allows the user to output a RADIOSS (.**rad**) data file directly from the VPG database. When writing a RADIOSS input file, VPG creates default materials and element properties for some elements. For specific translation properties, see the RADIOSS conversion tables, Appendix D, H.

1. VPG prompts:
   > ENTER FILE NAME OR "STOP" TO EXIT
   - Enter a file name (up to 24 characters).

2. If the file name already exists, the program will prompt the following message:
   > FILENAME ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - **YES** will overwrite the existing file.
   - **NO** will prompt for a file name.

3. VPG prompts:
   > SELECT RADIOSS VERSION
   ? 1 - VERSION 2.2
   2 - VERSION 2.3

4. VPG prompts:
   > ENTER TITLE
   - The user enters an alpha-numeric title containing a maximum of 48 characters.
   - **VPG** returns the user to the **FILE MANAGER** menu.

5. VPG prompts:
   > OUTPUT ONLY ACTIVE PARTS? (Y/N)
   - **YES** outputs only the elements displayed on the screen.
   - **NO** outputs all of the existing elements in the database.
   - After entering the appropriate selection, the user returns to the FILE MANAGER menu.

note: If the properties/materials are not assigned to the parts in the database, VPG prompts the user to define the properties/materials by default, by global default, or by part. Supported properties are listed in Appendix D, *VPG Capabilities for F.E.A. Programs.*

SUMMARY REPORT

This function allows the user to output a model summary with the following information:

1. the break down of node ranges by part
2 the break down of element ranges by part
3. the break down of part connectivity (part to part connections) by part.

The file is written in the working directory as an ASCII file.

PRE PROCESSOR

The Pre Processor menu contains a series of submenus with the ability to create and edit lines, surfaces, elements, material properties and element properties.

PRE PROCESSOR

LINE
SURFACE
ELEMENT
NODE
MODEL CHECKER
SET MENU
BOUNDARY CONDITION
CONSTRAINT
MATERIAL
ELEMENT PROPERTY
CONTACT INTERFACE
DYNA MISC
SUPER ELEMENT
EXIT

The Pre Processor submenus are described in their own individual sections which follow.

note: DYNA MISC only appears when the analysis code is set to LS-DYNA. SUPER ELEMENT only appears when the analysis code is set to NASTRAN.

LINE

The functions of the **LINE MENU** are intended to create a new set of line data or modify existing line data. VPG currently has a limitation of 150,000 lines or 200,000 points per database. The following options are available in the **LINE MENU**:

LINE

ADD POINTS
GENERATE ARCS
ARC TANGENT 2 LINES
3 POINT ARC
COPY
COMBINE LINES
SPLINE CURVE
DELETE
EXTEND LINE
GENERATE LINES
IDENTIFY
MIRROR
OFFSET
PROJECT
RESPACE LINE
REVERSE DIRECTION
SCALE LINES
SECTION LINE
SHOW
SPLIT
TRANSFORM
VIEW
EXIT

A detailed description of each function is given in the following sections.

ADD POINTS

This function allows the user to add points to an existing line either randomly or between two selected points.

1. VPG prompts:
   > SELECT LINE FOR ADDED POINTS

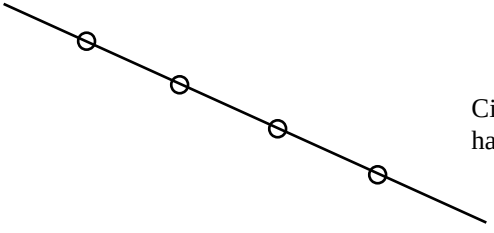
2. VPG prompts:
   > SELECT POINT LOCATION ON LINE
   - The user may create new points by placing the cursor anywhere on the line and pressing the left mouse button; or the user may select one the following options:

     BETWEEN 2 POINTS
     - This option allows the user to select 2 end points of a line segment and create a specified number of new points between the them (see step 4).

     CURSOR LOCATION
     POINT
     REJECT LAST

3. VPG prompts:
   > BETWEEN 2 POINTS
   > SELECT POINT 1 ON LINE
   > SELECT POINT 2 ON LINE
   > ENTER NUMBER OF POINTS TO BE ADDED
   - The user enters the desired number of points.

4. EXIT or DONE returns the user to the LINE menu.



Circles indicate the locations where points have been added.

Continued on next page.

GENERATE ARCS

This function allows the user to generate arcs and circles either globally, locally or about a selected node or point. The created arc will be included in the CURRENT PART (see PART CONTROL MENU).

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global) about which to create the arc. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in the *Local Coordinate System, Section 2.14.*

   note: Arcs and circles are generated about the local W-axis or global Z-axis.

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (Y/N/A)

3. If accepted, VPG prompts:
   > ENTER RAD, TH1, TH2, INC
   - RAD = radius of arc.
   - TH1 = beginning point of arc or circle.
   - TH2 = ending point of arc or circle.
   - INC = angle increment between points (default is 5 degrees).
   - Any real value greater than zero is valid. i.e., 100,0,360,10 is a circle with a radius of 100 units with an increment of 10 degrees between points.

   An arc and a circle have been generated with the same center.

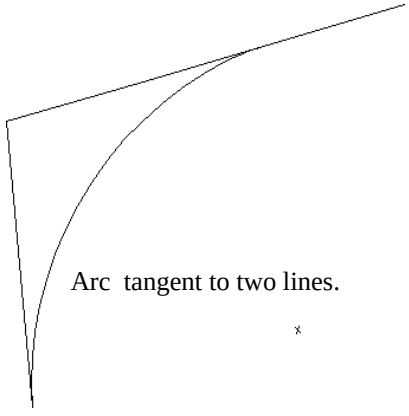
4. VPG prompts:
   > SAME CENTER AND PLANE? (YES/NO/EXIT)
   - Do you want to create another circle/arc using the same center and plane?
   - YES returns the user to step 3.
     NO returns the user to step 1.
     EXIT exits the function.

ARC TANGENT 2 LINES

This command allows the user to create an arc tangent to 2 lines. The arc will be included in the CURRENT PART (see PART CONTROL MENU). The two intersecting lines will be trimmed by the tangent arc.

1. VPG prompts for a line:
   > SELECT LINE 1

2. VPG prompts for a second line:
   > SELECT INTERSECTING LINE



Arc tangent to two lines.

3. VPG prompts:
   > ENTER RADIUS OF ARC OR -1 TO EXIT
   - Any real value greater than zero is valid.
   - An arc tangent to the 2 lines is drawn. The lines need to be long enough to make actual contact with the arc but do not need to intersect.

3 POINT ARC

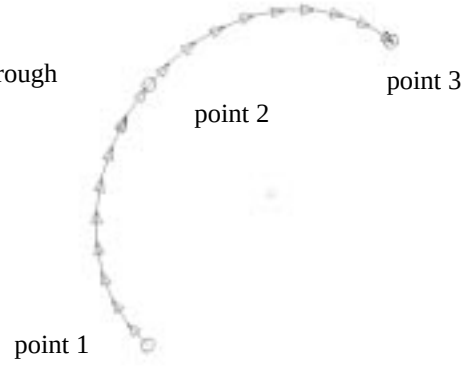
This function allows the user to create an arc through 3 points/nodes that are not colinear. The created arc will be included in the current part.

1. VPG prompts:
   > PICK NEXT POINT OR NODE
   - This command will be prompted 3 times.
   - When 3 points or nodes are selected an arc will automatically be drawn.

2. EXIT will terminate this function.

Continued on next page.

The circles represent the points through which the arc is drawn.



COPY

This function allows the user to generate a set of duplicate lines. The user may then simultaneously translate and/or rotate the copied lines.

1. VPG prompts:
   > PICK LINE TO COPY
   > SELECT LINES
   - Select DONE once desired lines have been selected.

2. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global). VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*
   - Once the desired coordinate system is acquired, it will be displayed on the screen.

3. VPG prompts:
   > ACCEPT? (Y/N/A)
   ? YES
   - If YES, then go to step 4.

Continued on next page.

NO

- If NO, then go to step 2.

ABORT

- If ABORT, then function is terminated.

4. VPG prompts:
   > ENTER NUMBER OF COPIES (positive integer)

5. VPG prompts:
   > MOVE OR ROTATE (M/R)
   ? MOVE
   ROTATE
   ABORT
   - If MOVE is entered,
     > ENTER U, V, W INCREMENTS
     - i.e., 0,0,1000 will copy the selected line(s) 1000 units along the W-axis.

     > INCLUDE COPIED LINE IN ITS ORIGINAL PART (Y/N)
     - Allows the user to specify the desired part.
   - If ROTATE is entered,
     > ENTER ANGLE INCREMENT (degrees)
     - i.e., 200 degrees.

     > INCLUDE COPIED LINE IN ITS ORIGINAL PART (Y/N)
     - Allows the user to specify the desired part.
   - If ABORT is entered, then the function is terminated.

COMBINE LINES

This function allows the user to combine multiple lines into a new single line with a new line number. The original lines will be deleted after they are combined. The combined line should be selected in a logical sequence to form the new line.

1. VPG prompts:
   > PICK LINES TO COMBINE

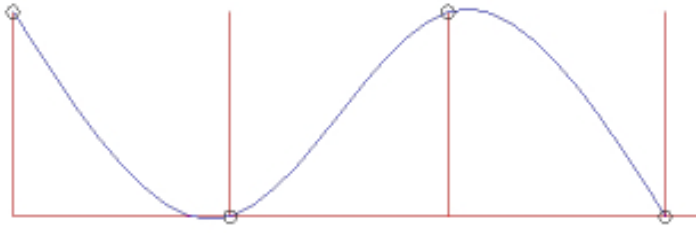
2. Once all the lines are selected, DONE or EXIT combines the lines and exits the command.

SPLINE CURVE

This function allows the user to draw a spline curve line through multiple points or nodes or any combination of points or nodes. A minimum of 3 points or nodes are required.

1. VPG prompts:
   > PICK NEXT POINT OR NODE
   ? ABORT
   DONE
   EXIT
   KEY IN XYZ
   - The user may enter up to 3 points globally to define a point or node.
     INCREMENTAL XYZ
   - The user may enter DX, DY, DZ from the previous point selected to define a point or node.
     NODE
     POINT
     REJECT LAST

2. DONE and EXIT will exit the function.



An example of a generated spline curve.

DELETE

This function allows the user to delete selected lines.

EXTEND LINE

This function allows the user to extend a line by a specified distance along the axis of the line at either end of the line.

1. VPG prompts:
   > SELECT LINE TO EXTEND

2. VPG prompts:
   > SELECT END POINT TO EXTEND LINE
   - Select the desired end point of the line to extend.

3. VPG prompts:
   > ENTER LENGTH TO BE EXTENDED
   - Input any real number.

4. Selecting UNDO will undo the last extend operation.

GENERATE LINES

This function allows the user to draw a line between nodes or points via a combination of keyboard and mouse selections. The created lines are incorporated into the CURRENT PART (or see PART CONTROL MENU).

1. VPG prompts:
   > PICK NEXT POINT OR NODE
   INCREMENTAL XYZ
   - The user may enter an additional DX, DY, DZ from the previous point or node selected to continue drawing the line.
     NODE
     POINT
     REJECT LAST

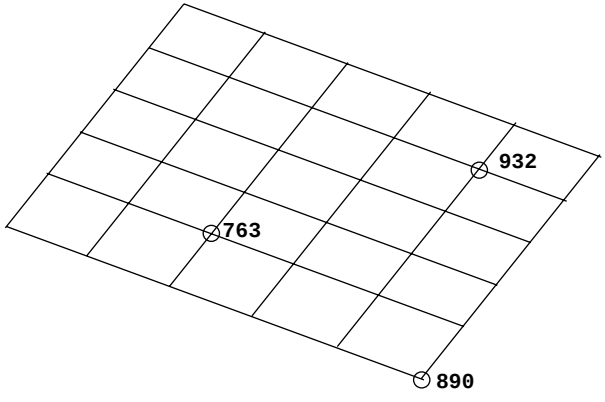
Continued on next page.

- DONE completes the creation of the line and starts the generation of a new line.
- EXIT will terminate this function.

IDENTIFY

This function allows the user to identify any node or point and its corresponding global location in X, Y, Z coordinates.

1. VPG prompts:
   > PICK NODES/POINTS
   ? ALL NODES
   EXIT
   KEY IN NODE #
   NODE
   POINT
   - If ALL NODES is selected, the numbers of the displayed nodes will appear on screen at their node locations (the first clear button in the DISPLAY PARAMETER OPTIONS WINDOW will remove the numbers)
   - If NODE is selected, the number of the node nearest the cursor location will appear on screen. The following message will echo on the command line:
     > NODE xxxx X= x.xxx Y= y.yyy Z= z.zzz
   - If POINT is selected, the number of the point nearest the cursor location will appear on screen. The following message will echo on the command line:
     > POINT xxxx X= x.xxx Y= y.yyy Z= z.zzz
   - EXIT will send the user to the LINE menu.



MIRROR

This function allows the user to reflect a copy of desired lines across a local or global coordinate plane.

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is now asking for a coordinate system to mirror the lines. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (Y/N/A)

3. If the user selects YES, then VPG prompts:
   > PICK LINES TO MIRROR
   > SELECT LINES
   > PICK A LINE(L), ELEM(C),
   > OR PART NAME(P) OF A PART.
   - The user must now select a target part with which to mirror lines.

4. Once desired lines have been selected, DONE or EXIT takes the user to the next step.

5. VPG prompts:
   > ENTER PLANE OF MIRROR (XY, YZ, XZ)
   - The user may now select a local plane of symmetry in which to mirror lines.

6. VPG prompts:
   > MIRROR LINES TO ORIGINAL PART? (Y/N)
   ? YES
   NO
   - YES will place all selected lines into the ORIGINAL PART.
   - NO will place all selected lines into the current part.

OFFSET

This function allows the user to copy and offset a selected line. The new line is generated in the u-v plane according to a user-specified distance and direction.

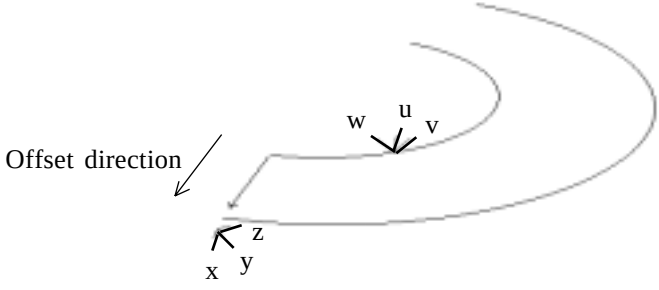
1. VPG prompts:
   > SELECT LINE TO OFFSET

2. Once the desired lines have been selected, VPG prompts:
   > DEFINE PLANE FOR LINE OFFSET
   > PICK NODE/PT FOR ORIGIN
   - The user enters a coordinate system to define the plane in which the line lies. If the line does not lie in the global XY, XZ, or YZ plane, then the user selects three points on the line to define the plane. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

3. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (YES/NO)

4. If the user selects YES, then VPG prompts**:**
   > ENTER OFFSET DISTANCE IN U-V PLANE OR -1 TO EXIT
   - The user enters a positive distance and an arrow appears at the beginning of the line.

5. VPG prompts:
   > ACCEPT OFFSET DIRECTION? (YES/NO)
   - YES creates a line in the displayed direction at the specified distance.
   - NO creates a line opposite the displayed direction at the specified distance.



PROJECT

This function projects lines and/or points onto selected surfaces.

1. VPG prompts:
   > SELECT SURFACES

2. Once the desired surfaces are selected, click on DONE. VPG prompts the user to create/choose a coordinate system.:
   > DEFINE PROJECTION VECTOR
   > PICK NODE/PT FOR ORIGIN

3. After accepting the coordinate system, VPG prompts:
   > SELECT ENTITY FOR PROJECTION:

4. Based on the selected entity, lines/points are projected onto the surface. If the point or points on the line do not intersect the surface along the global Z or W vector, VPG displays a message:
   > NO PROJECTION FOUND FOR XXXX POINT.

RESPACE LINE

This function allows the user to respace the number of points on the selected line equi-distantly.

1. VPG prompts:
   > SELECT LINE FOR RESPACING POINTS

2. Once the desired line is selected, VPG prompts:
   > ENTER NUMBER OF POINTS ON LINE OR E TO EXIT
   - Enter any integer greater than two. VPG includes the end points of the line as part of the total count of points on the line.

3. EXIT or DONE to exit this function.

REVERSE DIRECTION

Each line has a direction defined from the starting point to the end point of the line. This function allows the user to reverse the line direction. Arrowheads on the line (presented during SHOW LINE, SPLIT, etc.) are identifiable points and also indicate the line direction.

1. VPG prompts:
   > SELECT LINES
   - Once the line is selected the direction is automatically reversed.

SCALE LINES

This function allows the user to scale the selected lines or points with respect to any axis (global or local) using magnification factors in the X, Y, or Z (U, V, or W) directions. A scale factor greater than 1 will magnify the line, whereas a scale factor less than 1 will reduce the line.

1. VPG prompts:
   > PICK LINES TO SCALE
   > SELECT LINES
   > PICK A LINE(L), ELEM(C),
   > OR PART NAME(P) OF A PART
   - The user must now select a target part with which to scale lines.
   - Select **DONE** once the desired lines have been selected.

2. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG asks for a coordinate system to scale the lines and prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

3. Once the desired coordinate system is created, it will be displayed on the screen and the user will be prompted to accept or decline the system. If the user accepts, then VPG prompts:

Continued on next page.

> **ENTER X, Y, Z SCALE FACTORS**

- The user selects the desired scale factor.

note: The factors 1,-1,1 reflect the line size in the negative Y-direction (across the UV plane); the factors 0.5,0.5,0.5 reduce the lines by 0.5.

SECTION CUT

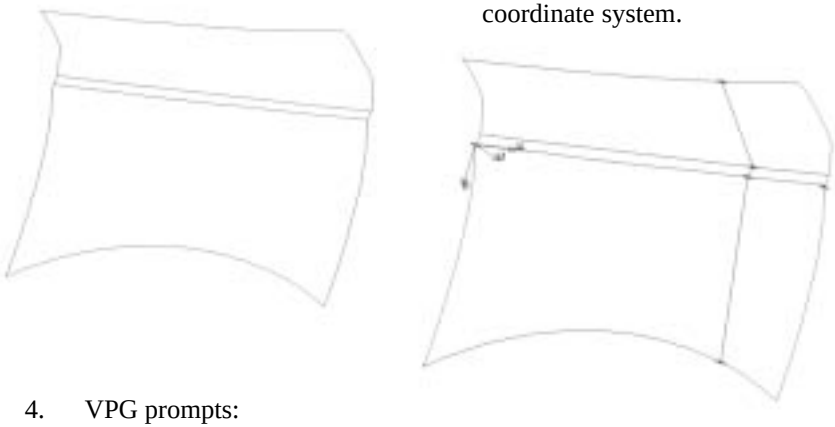
This function will generate line segments along a user defined UV plane at the intersection of selected lines intersecting the UV plane. The line section that is created will be included in the CURRENT PART (see PART CONTROL MENU).

1. VPG prompts:
   > **CREATE LOCAL COORDINATES**
   > **PICK NODE/PT FOR ORIGIN**
   - VPG is now asking for a coordinate system to create section lines. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

2. Once the desired coordinate system is acquired, it will be displayed on the screen.

3. If the user accepts the coordinate system, VPG prompts:
   > **DISTANCE FROM PLANE, W=?**
   - Users may now input a distance along the W-axis from origin of the selected coordinate system.
   - Any real value is a valid response.
   - The section line will be created at the distance W in the UV plane.

A section line is defined at a specific distance from the user defined local coordinate system.

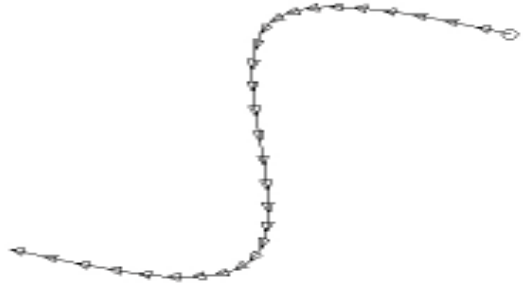


4. VPG prompts:
   > SELECT LINES
   - ABORT exits the function without creating a section line.
     DONE will complete the section line and return the user to step 4.
     EXIT completes the section line and exits this function.

SHOW LINES

The user may identify any existing line or line direction using this function. The selected line is highlighted (the starting point is labeled with a circle) and subsequent points are represented with arrowheads oriented in the direction of the line.

The arrows indicate point locations on the line and line direction.



SPLIT (LINE)

The user may split a line at a desired cursor location, at the nearest point, or at an intersection with another line.

1. VPG prompts:
   > PICK LINE TO SPLIT

2. VPG prompts:
   > PICK SPLIT LOCATION
   ? ABORT
   CURSOR LOCATION (anywhere the user clicks the mouse)
   LINE
   (the line will split at the location closest to the intersection of the selected line)
   POINT
   (the line will split at the selected point closest to the selected line)
   REJECT LAST
   - ABORT will exit this function.

TRANSFORM

This function allows the user to translate or rotate the selected lines or points to a new location.

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global). VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

2. Once the coordinate system is determined, VPG prompts:
   ? MOVE
   ROTATE

Continued on next page.

ABORT

- If MOVE is entered:

> ENTER U, V, W INCREMENTS

- i.e., 0,0,1000. This will move the selected line(s) 1000 units along the W-axis.
- If ROTATE is selected, the user enters an angle increment. Note: all rotational commands are about the local W-axis or the global Z-axis.

4. VPG prompts:

> SELECT ENTITY FOR TRANSFORM
? POINT
LINE
ABORT

5. If the user selects the point option to transform, the following options are available:

? AGAIN

- Repeats the previous transformation.

DONE
EXIT
LINE
REVERSE OPERATION

- Reverses the previous transformations.

6. If the user selects a line option to transform, VPG prompts the user to select a line or point.

7. After selecting the line/point, done executes the transformation.

SURFACE OPTIONS

Commands in this menu are used to create and modify cad surface data in VPG. VPG imports surface data and line data and converts it into VPG format with a bulletin. The IGES translator supports most IGES entities. Supported IGES entities are listed on page the following page.

SURFACE OPTIONS

BOUNDARY LINE
COPY
CREATE S-LINES
DELETE
2 LINE SURFACE
3 LINE SURFACE
4 LINE SURFACE
MIRROR
REVERSE NORMAL
RESPACE U-V LINE
SURFACE INTERSECT
U-V LINE ON/OFF
SCALE
SURF NORMAL ON/OFF
SPLIT
SECTION REVOLUTION
SHOW
SWEEP SURFACE
TRANSFORM
TRIM SURFACE
VIEW
EXIT

A detailed description of each function is given in the following sections.

note: The following are working definitions used throughout this section of VPG:

- **Domain**—The trimmed surface of the common region of the interior of the outer boundary and the exterior of the inner boundaries as well as the boundary curves.
- **Outer Boundary**—One boundary that is situated within the domain (the surface that is to be trimmed). In particular, it describes the boundary curve of the domain.
- **Inner Boundary**—Any number of inner boundaries including zero. The set of inner boundaries satisfies two criteria:

    (a) The curves as well as their interiors are mutually disjointed.
    (b) Each curve lies in the interior of the outer boundary.

SUPPORTED IGES ENTITY TYPES

VPG supports the following IGES entity types.

ENTITY NAME	TYPE	ENTITY NAME	TYPE
Null	0	Tabulated Cylinder Entity	122
Circular Arc	100	Transformation Matrix	124
Composite Curve	102	Rational B-Spline Curve	126
Conic Arc	104	Rational B-Spline Surface	128
Copious Data	106	Offset Surface	140
Plane	108	Boundary	141
Line	110	Curve on a Parametric Surface	142
Parametric Spline Curve	112	Bounded Surface	143
Parametric Spline Surface	114	Trimmed (Parametric) Surface	144
Point	116	Subfigure Definition	308
Ruled Surface	118	Singular Subfigure Instance	408
Surface of Revolution	120		

BOUNDARY LINE

This function creates a line on the boundary of a selected surface.

1. VPG prompts the user to select a surface on the screen.
   > SELECT SURFACE FOR BOUNDARY LINE

2. VPG creates a boundary line around the selected surface and includes it in the current part.

note: The boundary line should be split if it is intended for use with line meshing.

COPY

This function copies selected surface(s) to a new location.

1. VPG prompts the user to select surfaces to copy.

2. VPG prompts the user to create a local coordinate system. The user also has the option to select the global coordinate system. (Refer to *Local Coordinate System, Section 3.5* for a thorough explanation of this procedure.)

3. Once the desired coordinate system has been defined, VPG prompts for the number of copies.

4. Once the number of copies has been entered, VPG prompts for the method of copy.
   > MOVE OR ROTATE (M/R)
   ? MOVE
   > ENTER U, V, W INCREMENTS
   - i.e., /0,0,1000/. This copies the surface(s) 1000 units along the W-axis or global Z.

   ROTATE
   - If ROTATE is selected, enter the angle increment (right-hand rule) about
     the W-axis. The remaining commands are the same as the MOVE option.

   ABORT

Continued on next page.

5. VPG prompts:

```
>   INCLUDE COPIED SURFACE IN ITS ORIGINAL PART? (Y/N)
?   YES
    NO
```

CREATE S-LINES

This function creates sectional line data on the selected surface.

1. VPG prompts the user to select a surface on the screen.

2. Once the surface is selected, VPG places edge numbers on the surface and prompts the user to enter a number of section lines along edges 1 and 2.

3. VPG creates a set of section lines along edges 1 and 2. These lines are included in the current part.

DELETE SURFACES

This function deletes the selected surfaces from the database.

2 LINE SURFACE

This function creates a surface from two selected lines. The operation is similar to **2** LINE MESH for elements.

3 LINE SURFACE

This function creates a surface from three selected lines. The procedure is similar to **3** LINE MESH for elements.

note: Select lines in a clockwise or counterclockwise direction.
note: The direction of the surface normal dictates the direction of the plate element normals.

4 LINE SURFACE

This function creates a surface from four selected lines. The procedure is similar to 4 LINE MESH for elements.

note: Select lines in a clockwise or counterclockwise direction.
note: The direction of the surface normal dictates the direction of the plate element normals.

MIRROR

Reflects a copy of desired surfaces across a local or global coordinate plane.

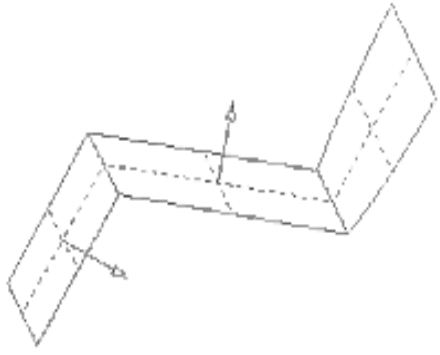
1. FEMB prompts the user to select a surface on the screen.

2. FEMB prompts the user to seelct a coordinate system followed by the plane of the mirror.

3. The user then has the option of putting the surface in a new part.

REVERSE NORMAL

This function reverses the normal direction of the surface. The surface normal affects light source shading on some workstations. REVERSE NORMAL also controls the normal direction of the plate elements during meshing.



RESPACE UV LINES

This function changes the UV line density of the selected surfaces.

1. VPG prompts the user to select a surface on the screen.

2. VPG labels the edge numbers on the surface and prompts the user to enter the number of UV lines along edges 1 and 2.

Continued on next page.

3. VPG creates a set of UV lines along edges 1 and 2. These UV lines are included in the current part.

note: This operation changes only the appearance of the selected surfaces. It does not change the integrity of the surface.

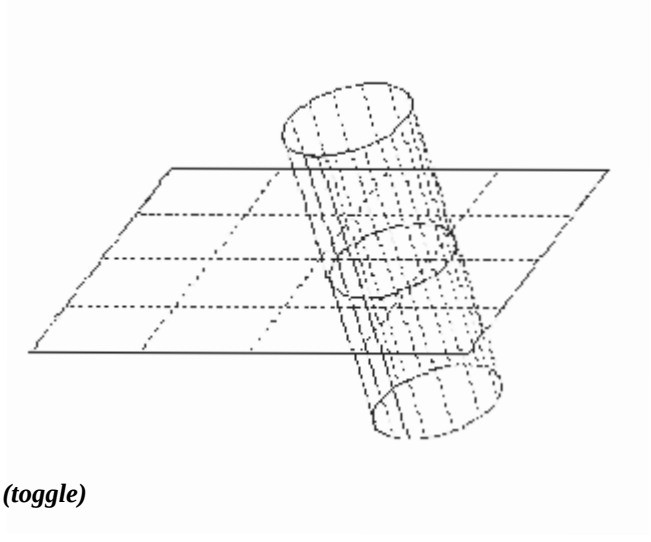
SURFACE INTERSECT

This function creates a line at the intersection between two selected surfaces.

1. VPG prompts the user to choose a surface.
   > SELECT THE FIRST SURFACE

2. After selecting the first surface, VPG prompts the user for the second surface.
   > SELECT THE SECOND SURFACE

3. After selecting the second surface, VPG automatically creates a line at the intersection of the two surfaces. The newly created line is included in the current part.



UV LINE ON/OFF (toggle)

This function controls the on and off display of UV lines on a surface.

note: Toggling the UV lines off has no effect on the outcome of meshing , trimming or splitting of surfaces.

SCALE

This function allows the user to scale selected surfaces with respect to any axis (global or local) using magnification factors in the X, Y, or Z (U, V, or W) directions. A scale factor greater than 1 will magnify the surface, whereas a scale factor less than 1 will reduce the surface.

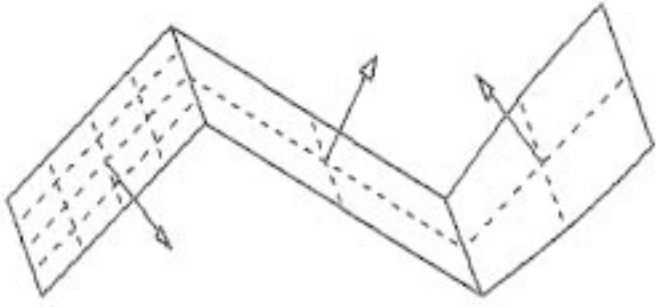
1. VPG prompts the user to select the surfaces.

2. Once the surfaces have been selected, VPG prompts the user to create a local coordinate system. The user also has the option to select the global coordinate system. (Refer to *Local Coordinate System, Section 3.5* for a thorough explanation of this procedure.)

3. Once the desired coordinate system has been defined, VPG prompts:
   > ENTER X,Y,Z SCALE FACTORS
   - This scales the surface.

SURF NORMAL ON/OFF (toggle)

This function controls the on and off display of normal vectors on a surface. The normal vector below is shown at the center of the surface.



SPLIT

This function allows the user to split selected surfaces in several different ways. All the options for splitting surfaces are addressed below.

1. VPG prompts for a surface.

2. Upon selection of the surface, VPG prompts :

   > SELECT OPTION TO SPLIT (DEFAULT IS SECTION LINE)

   • User selects a displayed section line to split a surface.

   ? LINEAR SEGMENTS

   > SELECT NEXT POINT FOR LINEAR SEGMENT

   • The user may pick up to 500 locations on the surface to complete a piecewise linear curve.

   RESPACE U-V LINES

   > ENTER THE NUMBERS OF UV LINES: N1, N2

   • N1 or N2 cannot be greater than 199.

   • The user may choose a desired section line, once the user selects the section lines for N1, N2 .

   SPLINE CURVE

   > SELECT NEXT POINT FOR SPLINE CURVE

   • The user may pick up to 500 locations on the surface in order to complete a spline curve.

   SURFACE INTERSECTION

   > SELECT THE SECOND SURFACE

   • Once the user selects the second surface, the first surface splits at the intersection of the two surfaces. The user may split the second surface at the same intersection.

   U-V LINES

   > SELECT UV LINE TO SPLIT SURFACE

   TWO POINTS ON BOUNDARY

   > SELECT FIRST POINT ON BOUNDARY

   > SELECT SECOND POINT ON BOUNDARY

   • VPG splits the surface once the user selects the second boundary point.

   EXIT

SECTION REVOLUTION

This function creates a surface by rotating a selected section line (the generatrix) up to 360º about a vector (the axis of revolution).

1. VPG prompts:
   > SELECT THE AXIS OF REVOLUTION
   > SELECT THE FIRST POINT
   ? ABORT

   LINE (TWO POINTS)
   - The user may select a line, defined by only two points, to be the axis of revolution.

   TWO POINTS
   - Default.

   REJECT LAST POINT
   UNDO

   - The user has been prompted to select the first of two points required to define the
   axis of revolution. Once the first point has been selected, VPG prompts:

   > SELECT THE SECOND POINT FOR THE AXIS

   - The user selects the second point, then VPG displays a vector representing the axis of revolution.

   > SELECT A LINE AS THE GENERATRIX OF REVOLUTION
   ? LINE (default)
   LINE SEGMENT (allows user to combine line segments into one line)
   DONE
   REJECT (rejects last selection)

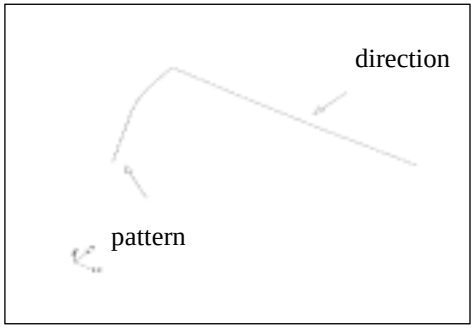
2. VPG prompts:
   > ENTER START AND END ANGLE [DEFAULT: 0., 360.]
   - The user may enter a desired start angle and end angle to generate a surface or press ENTER to generate a 360 degree angle.

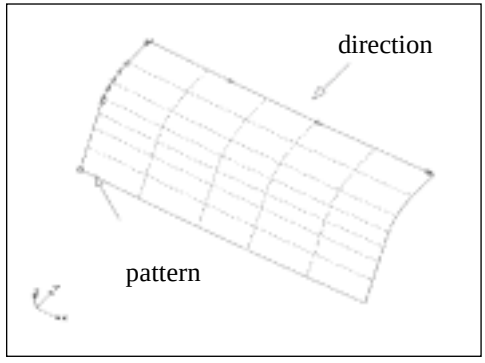
SHOW

This function highlights the selected surface and UV lines in white color. Choosing CLEAR in the VPG DISPLAY PARAMETER OPTIONS MODULE removes highlights.

SWEEP SURFACE

This function allows the user to create a surface via the selection of a section line and a direction line. The surface is generated along the length of the direction line.





1. VPG prompts the user to select a sweep type:
   > SELECT SWEEP TYPE
   NORMAL SWEEP
   - The section lines remain perpendicular to the line of direction.

   RIGID SWEEP
   - The section lines remain parallel to the line of pattern.

2. VPG prompts the user to select a surface.
   > SELECT A LINE FOR THE SWEEP PATTERN
   - The selected line will act as a constant cross section for the new surface.

3. Once the line has been selected, VPG prompts:
   > SELECT A LINE FOR THE SWEEP DIRECTION
   - The new surface will generate along the selected line and the user will return to step 2.

TRANSFORM

This function allows the user to move or rotate a selected surface either locally or globally.

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   VPG prompts the user to create a local coordinate system. The user also has the option to select the global coordinate system. (Refer to *Local Coordinate System, Section 2.14* for a thorough explanation of this procedure.)
2. Once the desired coordinate system has been defined, VPG prompts:
   > MOVE OR ROTATE (M/R)
   ? MOVE
   > ENTER U, V, W INCREMENTS
   - i.e. /0,0,1000/. This will transform the selected surface(s) 1000 units
     along the W-axis.

   ROTATE
   - If ROTATE is entered, enter angle increment (right hand rule) about the
     W-axis. The remaining commands are the same as the MOVE option.

   ABORT
   - User will now be prompted to select the surface to be transformed.
3. VPG prompts the user to select surfaces to transform. User also has the option to select AGAIN or REVERSE to repeat the rotation or movement. Once the desired surfaces are selected, enter DONE or EXIT to continue.

TRIM SURFACE

This function defines curves for trimming edges and/or cutting holes on the selected surface. The defined curves may be poly-lines, b-spline lines, circles, piecewise linear segments, or intersecting lines of two surfaces.

1. VPG prompts for a surface:
   > SELECT SURFACE FOR TRIM OPERATION

2. Upon selection of the surface, VPG prompts:

```
>   SELECT POINT ON SURFACE FOR TRIM LINE
    ABORT
    CIRCLE
    INTERSECTION LINE
    OPEN CURVE
    POINT
    SPLINE CURVE
    REJECT LAST POINT
    DONE (CLOSED)
    EXIT
```

This function cuts circles on a selected surface. The user defines the location of the circle via mouse pick.

-TRIM SURFACE
CIRCLE

1. VPG prompts:

```
>   SELECT THE CENTER OF THE CIRCLE
```

   - The user defines the center of the circle anywhere on the surface.

```
>   SELECT A POINT ON THE CIRCLE
```

   - The user may select a second point on the surface to define the radius of the circle. VPG proceeds to draw the circle.

2. The user should then select exit. VPG prompts:

Continued on next page.

> SELECT A CLOSED CURVE AS OUTER BOUNDARY
? ABORT
DONE
EXIT
SURFACE BOUNDARY
- Selects the outer, closed, surface boundary.

CURVE ON SURFACE
- The user may select an existing curve on the surface (open or closed)
  as the outer boundary of the trim operation.

REJECT LAST CURVE
- Deselects the curve selected for the trim operation.

3. VPG prompts:
   > SELECT CLOSED CURVE AS INNER BOUNDARY
   ? ABORT
   DONE
   EXIT
   NO INNER BOUNDARY
   - VPG trims the surface of the outer boundary.

   CURVE ON SURFACE
   - The user may select an existing curve on the surface (open or closed)
     as the inner boundary of the trim circle operation.

4. VPG then prompts:
   > DELETE THE ORIGINAL SURFACE? (Y/N)
   - YES generates a trimmed surface.
     NO generates a trimmed surface and retains a copy of the original surface.

5. The user is returned to step 1 of TRIM SURFACE command.

-TRIM SURFACE
INTERSECTION LINE

This function allows the user to trim away the intersection of two surfaces.

Continued on next page.

1. VPG prompts:
   > SELECT SURFACE FOR TRIM OPERATION
   note: This surface is referred to as a "primary surface" in this section.
   > SELECT THE SECOND SURFACE
   • Once the user selects the second surface, VPG draws a boundary line between the two intersecting surfaces.

   note: This surface is referred to as a "secondary surface" in this section.

2. Once the second surface has been selected and the user enters EXIT, VPG prompts:
   > SELECT A CLOSED CURVE AS OUTER BOUNDARY
   ? ABORT
   DONE
   EXIT
   SURFACE BOUNDARY
   • This creates the outer closed-surface boundary.
   CURVE ON SURFACE
   • This locates an existing curve on the surface (open or closed) as the outer boundary of the trim operation.
   REJECT LAST CURVE
   • This rejects the last curve selected for the trim operation.

3. VPG prompts:
   > SELECT CLOSED CURVE AS INNER BOUNDARY
   • This retains the surface between the outer boundary and the inner boundary during the trim operation. The inner boundary can be a line segment, open spline, closed spline, or circle.
   ? ABORT
   DONE
   EXIT
   NO INNER BOUNDARY
   • This trims the outside surface of the outer boundary.
   CURVE ON SURFACE
   • The user may select an existing curve on the surface (open or closed) as the inner boundary of the trim operation.
   REJECT LAST CURVE
   • This rejects the last curve selected for the trim operation.

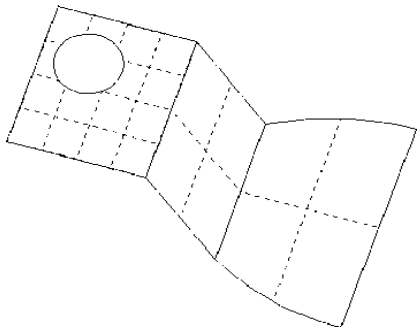
Continued on next page.

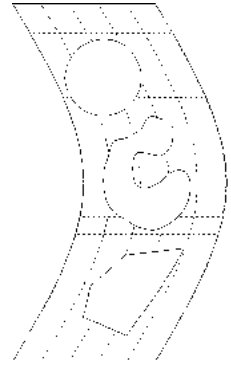
4. VPG then prompts:
   > DELETE THE ORIGINAL SURFACE? (Y/N)
   - YES generates a trimmed surface.
     NO generates a trimmed surface and retains the original surface.

5. VPG returns the user to step 2 of the TRIM SURFACE command.

-TRIM SURFACE
OPEN CURVE

This function defines an open curve on a selected surface. After determining, the surface, type of curve, and points, this function connects the points leaving the curve open, with the first and last point selected as the starting and ending points of the curve.





-TRIM SURFACE
POINT

POINT is the default option in the trim surface menu. Users may pick locations on the surface defining piecewise linear segments for trim operations .

-TRIM SURFACE
SPLINE CURVE

This function cuts spline curves on a selected surface. The user defines the location of the spline via mouse pick.

Continued on next page.

1. VPG prompts:
   > SELECT NEXT POINT FOR SPLINE CURVE.
   Ÿ Once the user has selected the points for the curve, then enters DONE (CLOSED) or DONE (OPEN) to complete the spline.
      - DONE (CLOSED) defines a closed spline curve passing through all the points selected.
      - OPEN CURVE defines a spline curve passing through all the points selected, with the first and last point selected as the starting and ending
        points of the curve. The user then selects exit twice.

2. VPG prompts:
   > SELECT A CLOSED CURVE AS OUTER BOUNDARY
   ? ABORT
   DONE
   EXIT
   SURFACE BOUNDARY
      - Selects the outer closed surface boundary.

   CURVE ON SURFACE
      - The user may select an existing curve on the surface (open or closed) as the outer boundary of the trim operation.

   REJECT LAST CURVE

3. VPG prompts:
   > SELECT CLOSED CURVE AS INNER BOUNDARY
   - The surface between the outer boundary and the inner boundary will be retained during the trim operation. The inner boundary can be a line segment, open spline, closed spline, or circle.

   ? ABORT
   DONE
   EXIT
   NO INNER BOUNDARY
      - The surface outside of the outer boundary will be trimmed.

   CURVE ON SURFACE
      - The user may select an existing curve on the surface (open or closed) as the inner boundary of the trim operation.

   REJECT LAST CURVE
      - Deselects the curve selected for the trim operation.

Continued on next page.

4. VPG then prompts:
   > DELETE THE ORIGINAL SURFACE? (Y/N)
   - YES generates a trimmed surface.
   - NO generates a trimmed surface and retains the original surface.

5. The user is returned to step 2 of TRIM SURFACE command.

-TRIM SURFACE
REJECT LAST POINT

This function deselects points when defining a trim line.

-TRIM SURFACE
DONE (CLOSED)

This function defines a closed curve on a selected surface. After determining, the surface, type of curve, and points, this function connects the points closing the curve, with the first and last points selected, linked.

ELEMENT OPTIONS

The ELEMENT OPTIONS menu contains the commands to create and modify elements, and to instantly mesh IGES surfaces. The 2, 3, and 4 LINE MESH commands create plate elements via line data selection, and the 6, 8, 9, and 12 LINE SOLID MESH commands create solid elements via line data selection. Elements can be modified to suit a particular model with the MODIFY, SPLIT, and COARSE ELEMENT commands.

VPG supports a wide range of element types for LS-DYNA and NASTRAN (refer to Appendix A *VPG Capabilities for F.E.A. Programs).* The commands in the ELEMENT OPTIONS menu are organized as follows:

ELEMENT OPTIONS

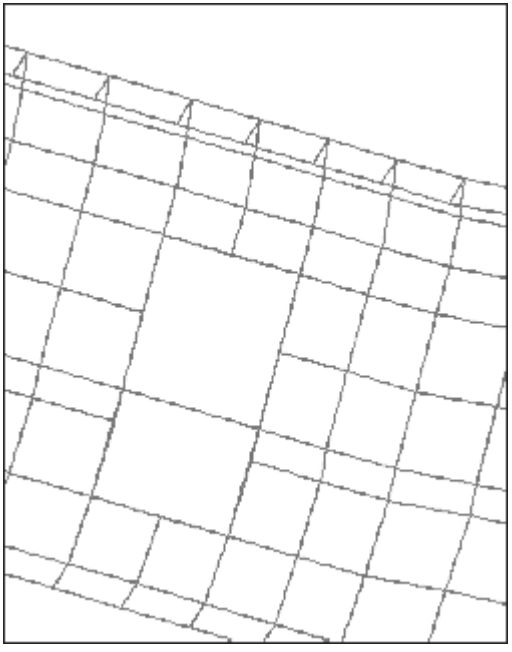
COARSE ELEMENT
CHANGE ID (NASTRAN only)
COPY
CREATE
DELETE
DRAG MESH
ELEMENT TYPE
IDENTIFY ELEMENT
ISO. SHELL FROM PLATE (NASTRAN ONLY)
LABEL
MIRROR
MODIFY
LINE MESH
RENUMBER
REVERSE NORMALS
SPLIT
SURFACE MESH
2 LINE MESH
3 LINE MESH
4 LINE MESH
6 LINE SOLID MESH
8 LINE SOLID MESH
9 LINE SOLID MESH
12 LINE SOLID MESH
TRANSFORM
ATTRIBUTE TABLE ON/OFF (LS-DYNA only)
VIEWING OPTIONS

COARSE ELEMENT

This function coarsens 2 CQUAD4 elements or 2 TRIA elements or a combination of a TRIA and a CQUAD4 into one element.

1. VPG prompts the user to select 2, 3, or 4 elements:

2. Once desired elements have been selected:
   - DONE coarsens the selected elements and prompts for more.
   - ABORT or EXIT will exit this function.



The larger elements were created from groupings of four smaller elements.

CHANGE ID

This function allows the user to change an existing element number. Parts do not need to be ON or CURRENT.

1. VPG prompts the user to select elements:
   > SELECT ELEMENT FOR NEW ELEMENT NUMBER

2. Once the user selects a desired element, VPG prompts the user to enter the new element number:
   > ENTER NEW EL. NO. FOR xxxx OR [CR] TO IGNORE
   - Where **xxxx** is the original element number.

3. If the user enters a number that already exists, the prompt will read:
   > ELEMENT NUMBER yyyy ALREADY EXISTS, REQUEST DENIED
   > ENTER NEW EL. NO. FOR xxxx OR [CR] TO IGNORE

note: This function only appears in NASTRAN analysis mode.

COPY

This function allows the user to generate a set of duplicate elements. The user may simultaneously translate, rotate, or offset (in the normal direction) the copied elements.

1. VPG prompts the user to select the elements to be copied:
   > SELECT ELEMENTS

2. Once the desired elements have been selected, DONE or EXIT continues to the next step where VPG prompts:
   > ENTER NUMBER OF COPIES

3. Then VPG prompts:
   > ENTER COPY OPTION (M/R/N/MA)
   ? MOVE
   ROTATE
   NORMAL OFFSET
   MAPPING (LCS) (see next command description)
   ABORT

Continued on next page.

4. VPG prompts:
   > CREATE LOCAL COORDINATES
   PICK NODE/PT FOR ORIGIN
   - VPG is asking for a desired coordinate system (local or global). VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

5. Once the coordinate system is determined, VPG prompts the user to select magnitudes for the transformation (respective to the selections made in step 4).
   - If **MOVE** was selected:
     > ENTER U, V, W, INCREMENTS
     - i.e., 0,0,1000. This will copy the selected element(s) 1000 units along the Z-axis.
   - If **ROTATE** was selected:
     > ENTER ANGLE INCREMENT
     - Enter angle increment (right hand rule) about the W-axis.
   - If **NORMAL** offset was selected:
     > ENTER THICKNESS IN NORMAL DIRECTION
     - Thickness is the distance away from the plane of the elements in the direction of the element's normal.

6. VPG prompts:
   > INCLUDE COPIED ELEMENTS IN ITS ORIGINAL PART? (Y/N)
   ? YES
   NO
     - YES places all copied elements in their original part.
     - NO places all copied elements into the CURRENT PART.

-COPY
MAPPING (LCS)

Mapping allows the user to copy and move or simply move selected elements from one user defined coordinate system to another user defined coordinate system along the UV plane.

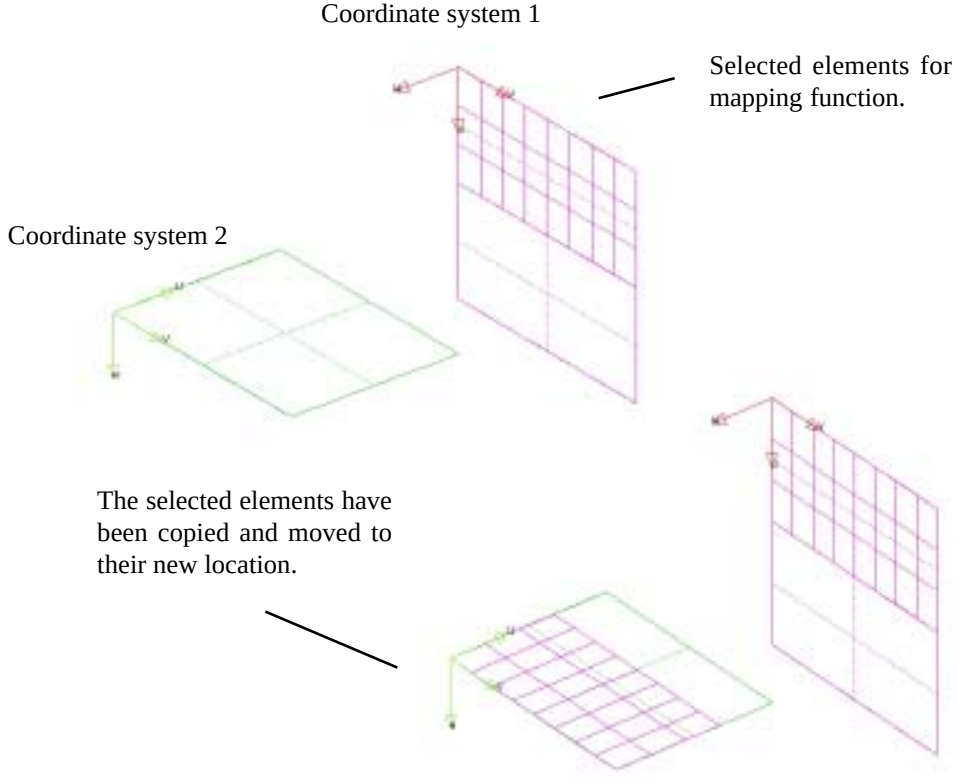
1. VPG prompts:
   > KEEP SELECTED ELEMENTS?

Continued on next page.

- YES copies the selected elements and moves the copied elements to the new location.
- NO removes the selected elements to the new location.

2. VPG prompts:
   > DEFINE FIRST LOCAL COORDINATE SYSTEM
   - This defines the orientation of the selected elements.

3. VPG prompts:
   > DEFINE SECOND LOCAL COORDINATE SYSTEM
   - This defines the new orientation for the selected elements.
   - VPG aligns the copied or moved elements according to their UV orientation.


Coordinate system 1
Selected elements for mapping function.
Coordinate system 2
The selected elements have been copied and moved to their new location.

CREATE

This function creates elements of specific type by connecting selected nodes or points. The element type refers to the analysis program defined at the creation of the database. Depending on the analysis program, NASTRAN or LS-DYNA, VPG offers a list of element types and prompts the user to select one. Because the NASTRAN element types are more comprehensive, they are documented first in the following pages with the LS-DYNA elements following.

ELEMENT EQUIVALENCY TABLE		
NASTRAN/ABAQUS	LS-DYNA / nonlinear codes	CMOLD/MOLDFLOW
BEAM ELEMENT (CBAR) **RIGID LINK (RBE2)** **PLOTEL ELEMENT** **PLATE ELEMENT** **SOLID ELEMENT** **SPRING (CELAS2)** **SPRING (CELAS1)** **RIGID BAR (RBAR)** **MASS (CONM2)** **BEAM (CBEAM)** **TRUSS ELEMENT** **HOLLOW TUBE** **ISOPARAMETRIC SHELL** **RIGID BODY (RBE3)** **SPOTWELD**	**BEAM** **NODAL RIGID BODY** **THICK SHELL** **PLATE** **SOLID** **SPRING/DAMPER** **MASS** **SEATBELT** **SPOTWELD** **JOINT** **INERTIA**	**PART ELEMENT** **CHANNEL** **HOT RUNNER** **COLD RUNNER** **CONNECTOR**

-CREATE
BEAM ELEMENT (CBAR) (NASTRAN)

This section covers the options for the BEAM ELEMENT subsection of CREATE ELEMENTS.

1. VPG prompts:
   > PICK AN ELEM(C) OR PROPERTY NAME(P) OF A PROPERTY

2. Once a beam property ID has been entered the prompt will read:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry or any combination of the three.

3. Once the desired nodes/points have been selected, VPG prompts:
   > PICK NODE/PT OR VECTOR FOR BEAM ORIENTATION
   ? NODE
   POINT
   ORIENTATION VECTOR
   - Up to three values may be entered to define a vector in the bar local XY plane. The local Y-axis of the bar will be plotted at the first node of the bar element as it is generated.

4. VPG prompts:
   > ENTER BEAM VECTOR
   - i.e., 0,0,1

-CREATE
RIGID LINK (RBE2) (NASTRAN)

This section covers the options for the RIGID LINK subsection of CREATE ELEMENTS.

1. VPG prompts:
   > ENTER INDEPENDENT D.O.F. (degrees of freedom)
   - Any combination of integers 1, 2, 3, 4, 5 and 6 may be entered as degrees of freedom.
   - Blanks or commas are not allowed between the degrees of freedom. i.e., 123456 = all D.O.F.

Continued on next page.

2. Next, VPG prompts for nodes/points for element:
   > PICK NODES/POINTS
   > AT INDEPENDENT NODE
   - To create the elements the user may select a node, point, keyboard entry or any combination of the three.

3. Once the desired node or point has been selected, VPG prompts:
   > AT DEPENDENT NODE
   - The user may select up to 200 dependent nodes.

4. Once the user picks the dependent node(s), DONE registers the selection and restarts the prompt cycle.

-CREATE
PLOTEL ELEMENT (NASTRAN)

This section covers the options for the PLOTEL ELEMENT subsection of CREATE ELEMENTS. PLOTEL elements are used as the cross-section pattern for the DRAG MESH operation.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three.
   - An element will be created after two nodes/points are selected.
   - EXIT or ABORT will exit this function.

note: PLOTEL elements may be output to a NASTRAN bulk data file. They are ignored for other analysis programs.

-CREATE
PLATE ELEMENT (NASTRAN)

This section covers the options for the PLATE ELEMENT (quadrilateral and triangular) subsection of CREATE ELEMENTS.

Continued on next page.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three.
   - A CQUAD4 ELEMENT requires 4 nodes or points to be selected.
   - A CTRIA3 requires 3 nodes or points to be selected, then an entry of DONE.
   - EXIT or ABORT will exit this function.

-CREATE
SOLID ELEMENT (NASTRAN)

This section covers the options for the SOLID ELEMENT (cube, wedge, tetrahedron) subsection of CREATE ELEMENTS.

note: A solid element cannot be created in a part that has plate properties. The user must either create a new part and assign it solid properties or make a part current that contains solid properties.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three:
     CTETRA: Pick 4 node points and type DONE.
     CPENTA: Pick 6 nodes/points (pick the second set of 3 nodes/points in the same order as the first set of 3 nodes/points) and type DONE.
     CHEXA: Pick 8 nodes/points (pick the second set of 4 nodes/points in the same order as the first set of 4 nodes/points). The element will be created.
   - EXIT or ABORT terminates this function.

-CREATE
SPRING ELEMENT (CELAS2) (NASTRAN)

This section covers the options for the SPRING ELEMENT (NASTRAN CELAS2) subsection of CREATE ELEMENTS.

1. VPG prompts:
   > ENTER SPRING DIRECTION (1 to 6) AT EACH END

Continued on next page.

- Two entries are required, one for each end of the spring (e.g., 1,1) for the "x" direction.
- Any number of spring elements may be created for the same connectivity depending upon the number of directions required.
  1 - translational along the global X-axis
  2 - translational along the global Y-axis
  3 - translational along the global Z-axis
  4 - rotational along the global X-axis
  5 - rotational along the global Y-axis
  6 - rotational along the global Z-axis

2. Once the desired direction has been entered for the spring, VPG prompts:
   > ENTER SPRING RATE
   - Any positive real number may be entered for the spring rate.

3. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three.
   - Select 2 nodes/points. The element will be created.
   - EXIT or ABORT terminates this function.

-CREATE
SPRING ELEMENT (CELAS 1) (NASTRAN)

This section covers the options for the general SPRING ELEMENT subsection of CREATE ELEMENT.

1. VPG prompts:
   > ENTER SPRING DIRECTION (1 to 6) AT EACH END
   - Two entries are required, one for each end of the spring (e.g., 1,1) for the "x" direction.
   - Any number of spring elements may be created for the same connectivity depending upon the number of directions required.
     1 - translational along the global X-axis
     2 - translational along the global Y-axis
     3 - translational along the global Z-axis
     4 - rotational along the global X-axis

Continued on next page.

5 - rotational along the global Y-axis
6 - rotational along the global Z-axis

- Once the desired direction has been entered for the spring, VPG prompts the user to enter a property. If no properties exist in the database at the current time VPG prompts to create one.

2. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry or any combination of the three.
   - Select 2 nodes/points. The element will be created.
   - EXIT or ABORT will exit this function.

-CREATE
RIGID BAR (RBAR) (NASTRAN)

This section covers the options for the RIGID BAR section of CREATE ELEMENT.

1. VPG prompts:
   > ENTER D.O.F. (CNA, CNB, CMA, CMB)
   - Any combination of integers 1, 2, 3, 4, 5 and 6 may be entered as degrees of freedom.
     1 - translational along the global X-axis
     2 - translational along the global Y-axis
     3 - translational along the global Z-axis
     4 - rotational along the global X-axis
     5 - rotational along the global Y-axis
     6 - rotational along the global Z-axis
   - Blanks or commas are not allowed between the degrees of freedom. i.e., 123,456,456,345

2. Once the desired D.O.F. has been entered the prompt will read :
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three.
   - Once the desired nodes (2) or points (2) have been selected the element will be created.
   - EXIT or ABORT will exit this function.

Continued on next page.

-CREATE
MASS (CONM2) (NASTRAN)

This subsection covers the options for the MASS subsection of CREATE ELEMENTS.

1. VPG prompts:
   > ENTER MASS VALUE AND MOMENTS OF INERTIA
   > SELECT AN ITEM TO EDIT
   - VPG displays a pop up window with the following items: I11, I21, I22, I31, I32, I33 etc. = MOMENT OF INERTIA
   - The user may enter only the mass if desired. The moment of inertia is an optional entry.
   - The user should remember that the units of the CONM2's should be consistent with the units of the database, e.g., DATABASE (MM.) = CONM2 (KG.)

2. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry or any combination of the three.
   - The elements will be created simultaneously upon selection of the desired node/points.
   - EXIT or ABORT will exit this function.

-CREATE
BEAM (CBEAM) (NASTRAN)

This subsection covers the options for the BEAM (NASTRAN CBEAM) section of CREATE ELEMENTS.

1. VPG prompts:
   > ENTER NEW PROPERTY NAME OR "EX" TO EXIT
   - VPG prompts the user to enter a property name if no properties exist in the database at the current time. Once a name is entered, a pop-up window will appear with a list of properties to edit.
   - This PID will overwrite the PID assigned for the shell elements for this part.

2. Once the beam property ID has been entered the prompt will read:

Continued on next page.

> PICK NODES/POINTS FOR ELEMENT

- To create the elements the user may select a node, point, keyboard entry, or any combination of the three.

3. Once the desired nodes/points have been selected, VPG prompts:

> PICK NODE/PT OR VECTOR FOR BEAM ORIENTATION

? NODE
POINT
ORIENTATION VECTOR

- Up to three values may be entered to define a vector normal to the beam local X-Y plane.

4. VPG prompts:

> ENTER BEAM VECTOR

- i.e., 0,0,1

-CREATE
TRUSS ELEMENT (NASTRAN)

This section covers the options for the TRUSS ELEMENT subsection of CREATE ELEMENTS.

1. VPG prompts:

> ENTER NEW PROPERTY NAME OR "EX" TO EXIT

- VPG prompts the user to enter a property name if no properties exist in the database at the current time. Once a name is entered, a pop-up window will appear with a list of properties to edit.

2. Once the property ID has been entered the prompt will read:

> PICK NODES/POINTS FOR ELEMENT

- To create the elements the user may select a node, point, keyboard entry, or any combination of the three.
- An element will be created after two nodes/points are selected.
- EXIT or ABORT will exit this function.

-CREATE
HOLLOW TUBE (NASTRAN)

This section covers the options for the HOLLOW TUBE subsection of CREATE ELEMENTS.

1. VPG prompts:
   > ENTER NEW PROPERTY NAME OR "EX" TO EXIT
   - VPG prompts the user to enter a property name if no properties exist in the database at the current time. Once a name is entered, a pop-up window will appear with a list of properties to edit.

2. Once the property ID has been entered the prompt will read:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three.
   - An element will be created after two nodes/points are selected.
   - EXIT or ABORT will exit this function.

-CREATE
ISOPARAMETRIC SHELL (NASTRAN)

This section covers the options for the ISOPARAMETRIC SHELL subsection of CREATE ELEMENTS.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - Generating isoparametric tri or quad elements requires the user to select three nodes/points per side (6).

-CREATE
RIGID BODY (RBE3) (NASTRAN)

This section covers the options for the RIGID BODY (RBE3) subsection of CREATE ELEMENTS.

Continued on next page.

1. VPG prompts:
   > ENTER COMPONENTS OF MOTION (123 etc.)
   - Defines the degrees of freedom.

2. VPG prompts:
   > PICK NODES/POINTS
   > AT REFERENCE NODE

3. VPG prompts:
   > ENTER WEIGHTING FACTOR WT 1 (REAL) OR "O" TO END
   > ENTER COMPONENTS WITH WEIGHTING FACTOR (123 ETC.)
   > PICK NODES POINTS
   > FOR NODE WITH WEIGHTING FACTOR .100E+01

4. Step three is repeated until the user selects DONE or ABORT.

-CREATE
SPOTWELD (NASTRAN)

This function allows the user to create local coordinate systems to be attached to the RBAR or RBE2 elements for the application of weld study. Two options are provided. The first option *creates* the RBAR and attaches the local coordinate systems at each node automatically. The second option attaches the local coordinate system automatically to the *existing* RBAR elements.

1. VPG prompts:
   > SELECT COORDINATE SYSTEM TYPE
   RECTANGULAR
   CYLINDRICAL
   SPHERICAL
   ABORT

2. VPG prompts:
   > SELECT LOCAL AXIS ALONG RBAR/RBE2
   U
   V
   W
   ABORT

Continued on next page.

3. VPG prompts:
   > SELECT OPTION TO CREATE SPOTWELD
   NODE/POINT
   RBAR
   EXIT

4. VPG prompts:
   > SELECT NODE/POINT FOR INDEPENDENT NODE
   - This step is repeated once and the spotweld is defined.
   - Once the spotweld is defined its coordinate system is displayed.

-CREATE
NODAL RIGID BODY (LS-DYNA)

This function defines a NODAL RIGID BODY element between multiple nodes. The first selected node serves as a reference point; all additional nodes are connected to this first node (usually the center of the rigid body) for display purposes.

1. VPG prompts:
   > SELECT FIRST NODE FOR RIGID BODY
   - The first node is selected.
   - Nodes can be selected by cursor pick, drag window, multi-point region or by part.

   > SELECT NODES FOR RIGID BODY
   - The second and subsequent nodes are selected until the user selects DONE.
   - The rigid body is displayed with double lines and an "R" alongsided.

-CREATE
THICK SHELL (LS-DYNA)

This section covers the options for the THICK SHELL subsection of CREATE ELEMENTS.

note: A solid element cannot be created in a part that has plate properties. The user must either create a new part and assign it solid properties or make a part current that contains solid properties.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - To create the elements the user may select a node, point, keyboard entry, or any combination of the three:
   - EXIT or ABORT terminates this function.

-CREATE
SPRING/DAMPER (LS-DYNA)

This function defines a spring/damper element between two nodes/points. It is recommended that the user select the ATTRIBUTE TABLE ON/OFF in the element options menu prior to creating a spring/damper. This will pop-up the ELEMENT DISCRETE CARD for user input.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - This step is repeated once.
   - When the second node/point is defined, VPG displays the SPRING/DAMPER element with a "D" next to it.

-CREATE
SEATBELT (LS-DYNA)

This function defines a seatbelt element between two nodes/points. Other seatbelt functions such as ACCELEROMETER, RETRACTOR, SENSOR, etc. are located in the DYNA MISCELLANEOUS menu.

Continued on next page.

1. VPG prompts:
   > PICK NODES/POINTS FOR ELEMENT
   - This step is repeated once.
   - When the second node/point is defined, VPG displays the SEATBELT element with an "S" next to it.

2. Once the seatbelt elements have been created it is recommended that the user enter the DYNA MISC. menu in the PRE PROCESSOR menu and select SEATBELT to define the accelerometer, pretensioner, retractor, sensor and slipring.

-CREATE
JOINT (LS-DYNA)

This function allows the user to create joints for DYNA analysis. All types of joints are supported. The procedure to create joints has been simplified to create the coincident nodes for the second rigid body. The program first prompts for the Relative Penalty Stiffness (default=1.0). The program next prompts the user to select the first part (rigid body with material type 20) for the joint. If no material is defined for the selected part, the program prompts a warning message: "NO MATERIAL ASSIGNED TO SELECTED PART". The program then prompts the user to select necessary nodes in the first rigid body to define the orientation of the joint. The requirements for each type of joint are listed in the following table:

Continued on next page.

JOINT TYPE	NUMBER OF NODES	ORIENTATION	IDENTIFIER SYMBOL
Spherical	1 (N1)	Center	SJ
Revolute	2 (N1,N3)	Axis of Rotation	RJ
Cylindrical	2 (N1,N3)	Axis of Cylinder	CJ
Planar	2 (N1,N2)	Normal of Plane	PJ
Universal	3 (N1,N3,N4)	Center, End Points of the Perpendicular Axes	UJ
Translational	3 (N1,N3,N5)	Direction & Plane of Translation	TJ
Locking	3 (N1,N3,N5)	Axis & Plane	LJ

The program then prompts the user to select the second part (rigid body with material type 20) for the joint. All the required coincident nodes on the second part will be created and added to the second rigid body as CONSTRAINED_EXTRA_NODES_NODEs.

DELETE

This function allows the user to delete elements using several different functions.

DRAG MESH

This function offers three methods to extrude elements from a cross-section of pattern elements.

1. VPG prompts:
   > SELECT DRAG MESH OPTION
   ? ONE LINE DRAG
   - Allows user to drag 1D elements into 2D elements (beam to plate) or 2D elements into 3D elements (plate to solid) along a selected line.

   NORMAL DRAG
   - Extrudes plate elements into solid elements normal to the selected plate elements.

   3 OR 4 LINE DRAG
   - Extrudes beam elements (to plate elements) or plate elements (to solid elements) along 3 or 4 selected lines.

   EXIT

The following sections cover the details for each command.

-DRAG MESH
ONE LINE DRAG

This section covers the options for the ONE LINE DRAG subsection of DRAG MESH. This function extrudes the selected elements along the selected line to form the higher order elements (i.e., 1D elements (PLOTEL, beams) are extruded to form plates and plate elements are extruded to form solid elements).

1. VPG prompts:
   > SELECT ELEMENTS

2. Then VPG prompts:
   > SELECT CONTROL LINE FOR DRAG MESH
   - Number of points on the selected drag line dictates the number of sections created.
   - Line direction must point in the direction of the intended mesh.

3. Upon line selection, mesh is created. VPG prompts:
   > ACCEPT DRAG MESH? (Y/N)

-DRAG MESH
NORMAL DRAG

This section covers the options for the NORMAL DRAG subsection of DRAG MESH. The normal drag function extrudes the selected plate elements in the normal direction to create solid elements.

1. VPG prompts:
   > SELECT ELEMENTS

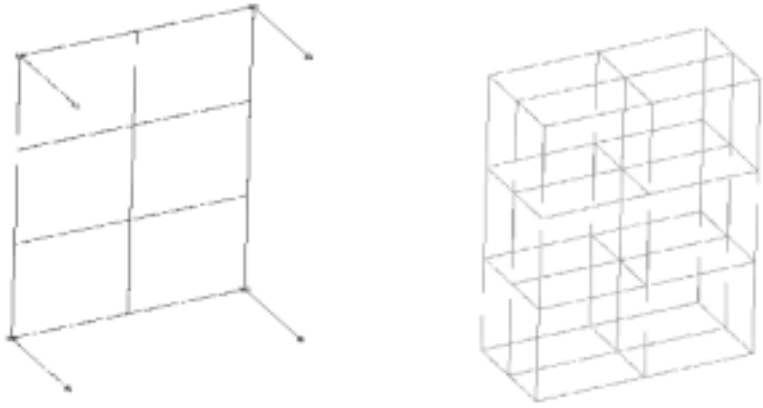
2. Then VPG prompts:
   > ENTER THICKNESS OF NORMAL DRAG
   - This is the total thickness of the drag operation.

   > ENTER NUMBER OF LAYERS THROUGH THICKNESS (DEFAULT 1)
   - This is the thickness of each layer of elements.

   > ACCEPT EXTRUDE MESH? (Y/N)

note: Plate element normals should be consistent prior to the execution of NORMAL DRAG.

Continued on next page.



-DRAG MESH

3 OR 4 LINE DRAG

This section describes the options for the 3 OR 4 LINE DRAG subsection of DRAG MESH. The line drag commands extrude the selected plate elements to create solid elements bounded by 3 or 4 lines in space.

1. VPG prompts:
   > SELECT ELEMENTS

2. VPG prompts:
   > SELECT 3 (THEN PRESS D) OR 4 CONTROL LINES
   - The user must select the 4 volume lines in clockwise or counterclockwise order.

3. Once the desired lines are selected, VPG prompts:
   > ENTER NUMBER OF LAYERS OF SOLID ELEMENTS
   ? ACCEPT MESH? (Y/N)
   - Line direction is not important during 4 LINE DRAG but the lines should be of reasonable length to form the desired shape. The control lines do not have to pass through the nodes of the plate elements.

ELEMENT TYPE

Defines the 2D element type prior to utilizing the 2, 3, 4, LINE MESH or SURFACE MESH.

1. VPG prompts:
   > PLATE ELEMENT TYPE
   ? QUADRILATERAL (default)
   TRIANGULAR
   ISO-PARAMETRIC (mid-side node)
   8 NODE THICK SHELL (available only in LS-DYNA)
   EXIT

IDENTIFY ELEMENTS

This function allows the user to identify an element and its nodes by cursor selection.

1. VPG prompts:
   > SELECT ELEMENT BY CURSOR
   - VPG identifies the element's respective type, number, part and nodes. The selected element will be highlighted, and element and node numbers will be displayed in the dialogue window.
   - The user also has the optoin to key in the number of a desired element to identify its location and characteristics.
   - It will also identify:

NASTRAN ELEMENT TYPE	DYNA ELEMENT TYPE	CMOLD/MOLDFLOW
CBAR	BEAM	PLATE
RBE2	RIGID EL	COLD RUNNER
PLOTEL	PLOTEL	HOT RUNNER
PLATE	PLATE	CONNECTOR
SOLID	SOLID	
CELAS2	SPRING	
CELAS1	SPOTWELD	
RBAR	MASS	
CTUBE	JOINT	
ISO-SHL	T-SHELL	
RBE3	SEATBELT	

ISO. SHELL FROM PLATE

This command converts 3- to 4-node plates into a 6- to 8-node isoparametric shell.

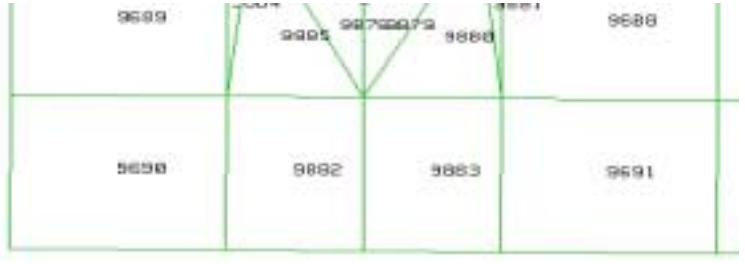
1. VPG prompts:
   > SELECT ELEMENTS
   - Once the desired elements are selected, DONE or EXIT will execute the command.

note: This function only appears in NASTRAN analysis mode.

LABEL

This function toggles the element label on or off. When LABEL is executed and parts are turned ON, the screen displays the elements and the element ID numbers in the database.

1. Depending on the current state of the toggle, VPG prompts either:
   > ELEMENT NUMBER WILL BE LABELED or
   > ELEMENT NUMBER WILL NOT BE LABELED
   - When ON, the element labels are superimposed on their respective elements.



The elements displayed with their element numbers.

MIRROR

This function allows the user to reflect a copy of desired elements across a local or global coordinate plane.

1. VPG prompts:
   > SELECT ELEMENTS

2. Once the desired elements have been selected, VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG asks for a desired coordinate system (local or global). VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

3. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (Y/N/A)
   - If the user selects NO, VPG returns to step 3.

4. If the user selects YES, VPG prompts:
   > ENTER PLANE OF MIRROR (XY, YZ, XZ)
   ? LOCAL UV PLANE
   LOCAL VW PLANE
   LOCAL UW PLANE

5. VPG prompts whether or not to keep the elements in their original parts:
   > INCLUDE MIRRORED ELEMENTS IN ITS ORIGINAL PART? (Y/N/A)
   ? YES
   NO
   - YES will place all selected elements into their source part.
   - NO will place all selected elements into the CURRENT PART.

Continued on next page.

MODIFY

This function allows the user to simultaneously delete and recreate any displayed elements. The displayed elements that are to be modified DO NOT need to be in the CURRENT part mode. VPG will automatically identify the element type of the selected element and prompt the user to recreate it. If the user selects a plate element, the prompt will read:

> MODIFYING PLATE ELEMENT
> PICK NODES/POINTS

Details are covered in **CREATE ELEMENTS** earlier in this section.

In LS-DYNA mode, MODIFY allows the user to edit element attributes (i.e. eid, pid, nid) if the ATTRIBUTE TABLE ON / OFF function in the ELEMENT OPTIONS menu is on. For further description, see the command ATTRIBUTE TABLE ON / OFF in this section.

note: For one-dimensional elements (beams, bars, etc.), VPG displays the selected beam's property ID and allows the user to modify it and the beam's connectivity.

LINE MESH

This function allows the user to generate one dimensional elements along a selected line automatically. The elements created will follow the direction of the selected line. The node sequence of the element begins at the starting point of the selected line according to the right hand rule. The created elements will be included in the CURRENT PART. (The CURRENT PART should be verified before proceeding with this function.)

1. VPG prompts:
   > SELECT ELEMENT TYPE
   - A list of available element types will be displayed.

2. If an element type that requires a property is chosen, then VPG prompts:
   > PICK AN ELEM(C) OR PROPERTY NAME OF A PROPERTY

3. VPG then prompts:
   > SELECT LINE FOR LINE MESH

Continued on next page.

4. Once the line selection has been made, VPG prompts:
   > ENTER NUMBER OF ELEMENTS ON SELECTED LINE
   - Any integer greater than 1 is valid.

5. VPG then prompts for the beam orientation:
   > PICK NODE/PT OR VECTOR FOR BEAM ORIENTATION

6. VPG prompts:
   > ACCEPT MESH (YES/NO/REMESH)
   ? YES
   NO
   REMESH
   - **YES** will accept mesh and prompt for more lines.
   - **NO** will not accept mesh or prompt for more lines.

7. ABORT, DONE, or EXIT exits this function.

RENUMBER

This function allows the user to renumber all elements in the existing database with the option of creating a summary table report of the element ranges between each part. The user may renumber elements in sequence or by parts.

1. VPG prompts:
   > OUTPUT ELEMENT NUMBER SUMMARY TABLE (Y/N)?
   ? YES
   NO
   - YES will prompt:
     > ENTER FILE NAME
     - Enter a file name into which to place the summary report.
   - NO will go to step 2.

2. VPG prompts:
   > ENTER RENUMBERING OPTION:
   ? EXIT
   SEQUENCE

Continued on next page.

> ENTER STARTING ELEMENT NUMBER (DEFAULT = 1)
PARTS
> ASSIGN STARTING ELEMENT NUMBER FOR EACH PART? (Y/N)
? YES

> ENTER STARTING ELEMENT NO. FOR PART xxxx.

- Once a number has been entered the prompt will read;

> ELEMENT x TO xx ASSIGNED, NEXT ELEM. NO.: yyyy

- If "blank" or "zero" is entered, yyyy will be used as the starting element number for the next part.
- These prompts will continue for every part in the database.

NO

> ENTER STARTING ELEMENT NUMBER (any positive integer)
> ENTER ELEMENT INCREMENT BETWEEN PARTS (1000)

- Any positive integer may be entered.

REVERSE NORMALS

This function allows the user to reverse the element orientation (normal) for all elements except CORD2R and CONM2.

1. VPG prompts:
   > SELECT ELEMENTS

2. Once desired elements have been selected, DONE or EXIT executes the command.

SPLIT

This function divides CQUAD4 and CTRI3 elements into 2 or 4 elements.

Continued on next page.

1. VPG prompts:
   > SELECT OPTION FOR SPLIT ELEMENT
   ? 2 QUADS (splits a QUAD4 into two QUAD4s)
   4 QUADS (splits a QUAD4 into four QUAD4s)
   2 TRIAS (splits a QUAD4 into two TRIAS)
   TRIA 2 TRIAS (splits a TRIA3 into two TRIA3s)
   TRIA QUAD COMBINE (splits a QUAD into a QUAD and a TRIA)
   TRIA 4 TRIAS (splits a TRIA into four TRIAs)
   ARBITRARY LOCATION (user defines locators for splitting element)
   EXIT

2. VPG prompts:
   > SELECT ELEMENTS TO SPLIT
   ? CURSOR LOCATION
   DONE
   EXIT
   MULTI-POINT REGION
   BY PART
   REVERSE OPERATION
   - Reverses the split direction. This option does not apply to group operations.

   UNDO LAST
   DRAG WINDOW

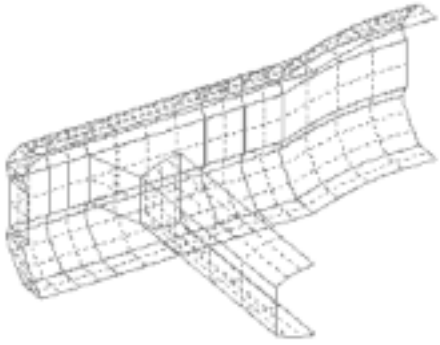
SURFACE MESH

This function allows the user to automatically mesh surface data. Currently VPG automeshes using 2D elements (plate elements). Supported surface types are covered in *Pre Processor, Section 2.1,* of this manual. The user has the option of automeshing by selecting one surface, multiple surfaces, or all of the displayed surfaces.

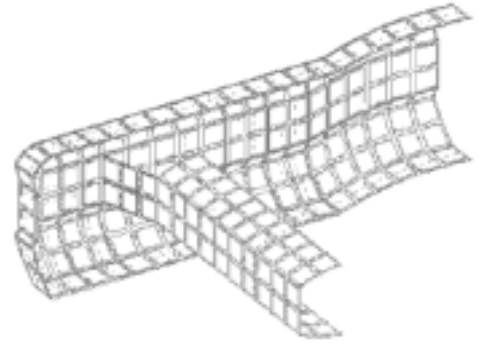
1. DISPLAYED SURFACES allows the user to automesh all surfaces at once. To model one surface at a time (default), the user selects an individual surface with the cursor and selects DONE. VPG prompts:
   > INCLUDE ELEMENTS IN THEIR ORIGINAL PART? (Y/N)
   - YES - VPG creates and stores the elements in the PID of the surface from which they were created.

Continued on next page.

- NO - VPG creates and stores the elements in the current part.

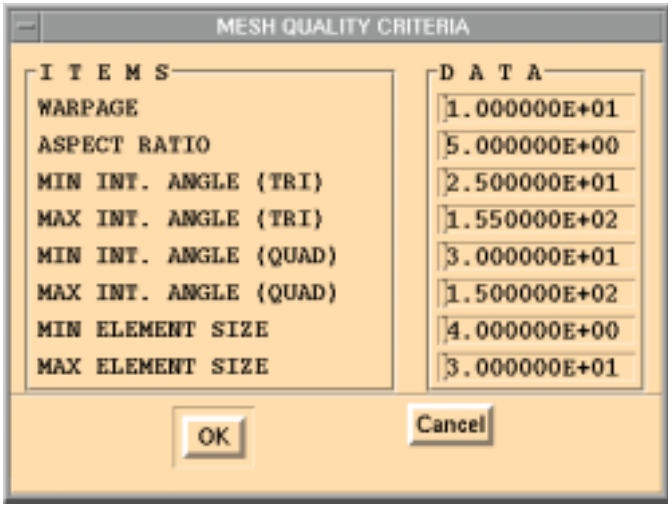


Surfaces to be meshed.



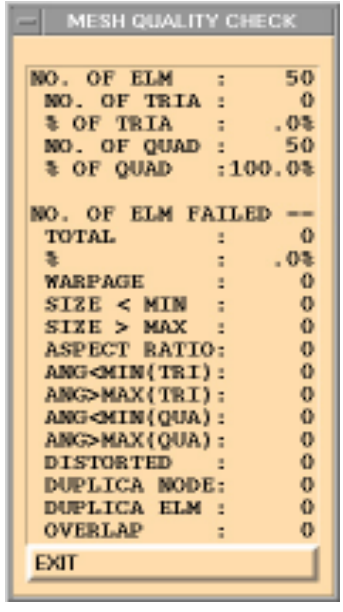
Surfaces after meshing.

2. After entering YES or NO, a pop-up window appears and VPG prompts:
   > DEFINE / MODIFY MESH QUALITY CRITERIA



3. VPG then prompts:
   > FINDING COMMON EDGE NODES OF SELECTED SURFACES
   > ENTER NUMBER OF ELEMENTS: N1, N2, N3, N4
   - See 2 LINE MESH command in this section for a description of this prompt.
   - If a trimmed surface is selected, only N1, N2 will be prompted.

4. Once the number of elements has been entered, VPG checks the mesh quality and displays the MESH QUALITY CHECK window. VPG prompts:
   > ACCEPT MESH? (YES/NO/RE-MESH)
   - VPG creates a mesh that merges coincident nodes on the boundary lines between surfaces. VPG optimizes most boundary problems.



5. To model multiple surfaces at once, users select multiple surfaces in the same way elements are selected.
   VPG prompts:
   > ENTER THE ELEMENT SIZE (global element size)

2 LINE MESH

This function generates elements between two selected lines. A rectangular pattern of N1 by N2 quadrilateral elements will be generated where N1 is the number of elements along the direction of the first selected line, and N2 is the number of elements between the two lines. For transition mesh, it is possible to enter N3 and N4 parameters for the mesh (each side may have a different number of elements e.g., 10, 5, 8, 7).

The nodes in the direction of N2 are generated by linear interpretation between the opposing nodes on the two lines. A row of N2 quadrilateral elements are generated first from the starting end of line 1 to the starting end of line 2. The pattern will be repeated N1 times along line 1. The node sequence starts at the beginning of line 1 and follows the direction of line 1 with the right hand rule. The created elements are included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

1. VPG prompts:
   > PICK LINE
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 2 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 2 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.

2. Once the user selects the desired lines the prompt will read:
   > ENTER NO. OF ELEMENTS ON EACH SIDE N1, N2, N3, N4
   - **N1** - any positive integer, greater than one for the number of elements along the selected lines.
   - **N2** - any positive integer, for the number of elements between the 2 lines.
   - **N3** - (optional) an integer value for the side opposite N1 where N1 is not greater than 2*N3 or N3 is not greater than 2*N1.
   - **N4** - (optional) an integer value for the side opposite N2 where N2 is not greater than 2*N4 or N4 is not greater than 2*N2.
     - i.e.,10,10 or 10,10,11,14 = ACCEPTABLE
     - i.e., 5,5,12,6 = NOT ACCEPTABLE

Continued on next page.

3. Once the desired element N1, N2, N3, N4 is entered, VPG prompts:
   > ACCEPT MESH (YES/NO/REMESH)
   ? YES
   NO
   REMESH
   - YES will accept the mesh and prompt for more lines.
   - NO will not accept mesh or prompt for more lines.
   - REMESH rejects the mesh and prompts for new N1, N2, N3, N4.

3 LINE MESH

This function generates elements in an area defined by 3 lines (line direction is not important). The lines defining the area do not have to intersect (an open area is possible). Elements are included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function. Two mesh patterns are available for 3 line mesh (triangular shape). VPG generates the triangular elements and allows the user to position them on the model.

1. VPG prompts:
   > SELECT 3 LINES
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 3 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 3 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.

2. Once the user selects the desired lines the prompt will read:
   > ENTER MESH TYPE: 1 OR 2?
   ? 1 - TRIANGLE AT CORNER
   - Triangles are concentrated in the vertex of the first two selected lines.
     > ENTER DIVISIONS N1, N2
     2 - TRIANGLES ALONG EDGE
   - Triangles are concentrated along the third selected line.
     > ENTER NUMBER OF ELEMENTS ON EACH SIDE?
     - VPG accepts only one entry as each side has the same number of elements.

4 LINE MESH

This function generates elements in an area defined by 4 selected lines. A rectangular pattern of N1, N2, N3 by N4 quadrilateral elements will be generated where N1 is the number of elements along the direction of the first selected line, and N2 is the number of elements along the direction of the second selected line, etc. For a transition mesh, it is possible to enter N3 and N4 parameters for the mesh (each side has a different number of elements e.g., 10, 5, 8, 7).

The nodes in the direction of N2 are generated by linear interpretation between the opposing nodes on the two lines. A row of N2 quadrilateral elements is generated from the starting end of line 1 to the starting end of line 2. This pattern is repeated N1 times along line 1. The node sequence starts at the beginning of line 1 and follows the direction of line 1 with the right hand rule. The created elements are included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

1. VPG prompts:
   > SELECT LINES
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 4 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 4 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.

2. Once the user selects the desired lines the prompt will read:
   > ENTER NO. OF ELEMENTS ON EACH SIDE N1, N2, N3, N4
   - N1 - any positive integer, greater than one for the number of elements along the selected lines.
   - N2 - any positive integer, for the number of elements between the 2 lines.
   - N3 - (optional) an integer value for the side opposite N1 where N1 is not greater than 2*N3 or N3 is not greater than 2*N1.
   - N4 - optional) an integer value for the side opposite N2 where N2 is not greater than 2*N4 or N4 is not greater than 2*N2.
     - i.e.,10,10 or 10,10,11,14 = ACCEPTABLE

Continued on next page.

3. Once the desired element N1, N2, N3, N4 is entered, VPG prompts:
   > ACCEPT MESH (YES/NO/REMESH)
   - YES will accept mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.
   - REMESH will not accept mesh and prompt for new N1, N2, N3, N4.

6 LINE SOLID MESH

This function is intended for automatic 6 or 8 node solid element generation in a volume. The volume is defined by top and bottom surfaces which are enclosed by 3 lines each. The lines must previously be defined (line direction is not important). The lines defining the area do not have to intersect (an open area is possible). The created elements will be included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

1. VPG prompts:
   > SELECT 3 LINES AT BOTTOM
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 6 LINE MESH function as in the COMBINE LINE function in the LINE section. The user must select this option before selecting the desired lines for 6 LINE MESH.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in the LINE section.

2. Once the user selects the desired lines the prompt will read:
   > SELECT 3 LINES AT TOP
   - Select the second set of three lines in the same order as the first set of three lines.

3. VPG prompts:
   > ENTER MESH TYPE: 1 OR 2 (1 or 2 will prompt;)
   1. **> ENTER NUMBER OF ELEMENTS N1 X N2 X N3**
   2. **> ENTER NUMBER OF ELEMENTS ALONG THE FACE**
      > ENTER NUMBER OF ELEMENTS BETWEEN THE FACES

4. Once **N1, N2,** and **N3** have been entered, the prompt will read:
   > ACCEPT MESH (YES/NO/REMESH)
   - YES will accept mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.
   - REMESH will not accept mesh and prompt for new N1, N2, N3.

8 LINE SOLID MESH

This function is intended for automatic 8-node solid element generation in a volume. The volume is defined by a top and bottom surface enclosed by 4 lines. Each line must be defined beforehand (line direction is not important). The lines defining the area do not have to intersect (an open area is possible). A row of N2 elements will be generated from the starting end of line 1 along line 4 on the bottom surface. The pattern will be repeated N1 times along line 1 on the bottom surface. The element pattern on the bottom surface will be repeated N3 times toward the top surface. The node sequence starts at the beginning of line 1 and follows the direction of line 1 using the right hand rule. The created elements will be included in the CURRENT PART. The CURRENT PART should be verified before proceeding with this function.

1. VPG prompts:
   > SELECT LINES FOR BOTTOM
   - LINE SEGMENT allows the user to combine a broken line while selecting lines for the 8 LINE SOLID function as in the COMBINE LINE function in the GEOMETRY BUILDER menu. The user must select this option before selecting the desired lines for 8 LINE SOLID.
   - The user may check the line(s) prior to executing this function by using SHOW LINE in GEOMETRY BUILDER.

2. VPG prompts:
   > SELECT LINES FOR TOP

3. Once the user selects the desired lines the prompt will read:
   > ENTER NO. OF ELEM. N1, N2, N3
   N1 - any positive integer, greater than one for the number of elements along the selected lines.
   N2 - any positive integer, for the number of elements between the 2 lines.
   N3 - any positive integer, for the number of elements between the bottom and top surface.

4. Once **N1, N2,** and **N3** have been entered, the prompt will read:
   > ACCEPT MESH (YES/NO/REMESH)
   - YES will accept mesh and prompt for more lines.
   - NO will not accept mesh and prompt for more lines.
   - REMESH rejects the mesh and prompts for new N1, N2, N3.

9 LINE SOLID MESH

This function allows the user to generate 6 or 8 node solid elements automatically in a volume. The volume is defined by 9 lines--3 lines for the top surface, 3 lines for the bottom surface (see 6 LINE MESH), and 3 lines connecting the top and bottom surfaces.

1. VPG prompts:
   > SELECT 3 LINE AT BOTTOM
   > SELECT 3 LINE AT TOP
   > SELECT 3 LINES ON THE SIDE, STARTING BETWEEN LINE 1 & 3
   - 9 LINE MESH is similar to 6 LINE MESH with the exception of 3 connecting lines.
   - Line direction (clockwise or counter clockwise) for all line selections should be consistent.

2. See 6 LINE MESH for further details and the prompt descriptions.

12 LINE SOLID MESH

This function allows the user to generate 8 node solid elements automatically in a volume. The volume is defined by 12 lines--4 lines for the top surface, 4 lines for the bottom surface (see 8 LINE MESH), and 4 lines for connecting the top and bottom surfaces.

1. VPG prompts:
   > SELECT 4 LINES AT BOTTOM
   > SELECT 4 LINES AT TOP
   > SELECT 4 LINES ON THE SIDE, STARTING BETWEEN LINE 1 & 4
   - 12 **LINE MESH** is similar to 8 **LINE MESH** with the exception of 4 connecting lines.
   - The user should maintain the consistent direction (clockwise, counter clockwise) for all line selections.

2. See section 8 LINE MESH for further details and prompt descriptions.

TRANSFORM

This function allows the user to transform selected elements by move, rotate, mapping (LCS) and normal offset. The normal offset option only applies to plate elements. It should be noted that the normal of the plate elements should be consistent for a proper normal offset.

1. FEMB first prompts the user to select the desired elements.

2. FEMB then prompts the user to select the desired transformation:
   MOVE moves the selected elements to a new location according to the distance entered along the selected axis.
   ROTATE rotates the selected elements to an angle increment (right-hand rule).
   NORMAL OFFSET moves the selected plate elements in the normal direction by the specified thickness.

3. FEMB prompts for a desired coordinate system (local or global) (this step is not required for NORMAL OFFSET). A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 2.14.*

4. FEMB prompts for the transformation value.

5. The elements are transformed.

6. FEMB prompts the user to select one of the following options:
   AGAIN performs the same transformation again with the same specifications.
   NEW TRANSFORM begins another transformation operation.
   REVERSE rejects the previous transformation.
   EXIT exits the function.

ATTRIBUTE TABLE ON/OFF

This function, available only in LS-DYNA mode, allows the user to add additional properties to elements that VPG does not prompt for when the elements are created (thickness, additional nodes, etc.) The ATTRIBUTE TABLE supports BEAM, DISCRETE, MASS, SEATBELT, SHELL, TSHELL and solid elements. It does not support JOINT, NODAL RIGID BODY, PLOTEL, and SPOT WELD elements.

NODE OPTIONS

The functions in the NODE OPTIONS menu relate to the two types of nodes that are defined in the VPG program. The first type are referenced nodes (connected by elements). The second type, the unreferenced nodes (not connected by any element), are represented with asterisks (*). Useful applications in this section include: moving nodes (node to node or node to point), checking for and merging common or coincident nodes, and transforming or rotating nodes.

NODE OPTIONS

NODE BETWEEN PT/NODE
CHECK DUPLICATE ID
CHANGE ID
COINCIDENT CHECK
COPY
COMPACT NODE ID
CREATE
DELETE UNREF. NODES
DISTANCE
IDENTIFY NODE/POINT
LABEL
MOVE NODE TO PT/NODE
PART CONNECTION
PROJECT
RENUMBER
SCALE
TRANSFORM
VIEW
EXIT

A detailed description of each function is given in the following section.

NODES BETWEEN PT/NODES

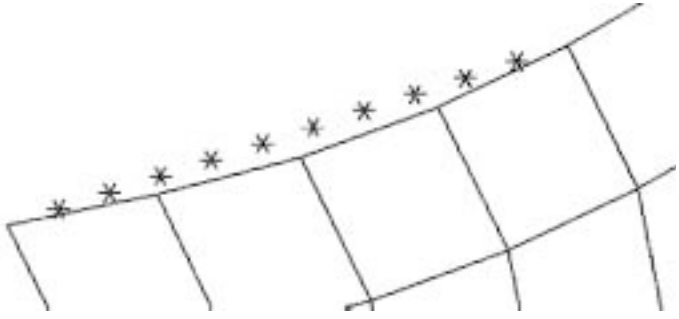
This command generates nodes that are spaced equally between two existing selected nodes or points. The new nodes are displayed as unreferenced (free) nodes and are labeled with asterisks (*).

1. VPG prompts for each point/node:
   > SELECT NODE/POINT 1
   NODE
   POINT
   REJECT LAST
   EXIT
   - Returns the user to the NODE OPTIONS menu.
   - The user can select the node nearest the cursor by pressing the left mouse button instead of typing the letter (N).
   - And then-

   > SELECT NODE/POINT 2

2. VPG prompts:
   > ENTER NUMBER OF NODES BETWEEN TWO POINTS
   - Any integer, one or greater, is valid.

3. The user has the option to enter EXIT to terminate.



Asterisks represent unreferenced nodes that have been created between two existing nodes.

CHECK DUPLICATE ID

This function renumbers any duplicate node numbers found in the VPG database. Users are unable to create duplicate nodes numbers from within the program, however, users may import files containing duplicate nodes from other programs. VPG will renumber the duplicates according to the value of the “maximum node number + 1."

1. If any duplicate nodes are found, the following messages echo on screen and in the VPG.**msg** user files located in the work directory:

   > CHECKING xxxx NODES FOR DUPLICATE NODES
   > DUPLICATE NODE xxxx CHANGED TO yyyy
   > xxxx DUPLICATE NODES FOUND

   • If no duplicate nodes are found, the following messages echo on screen and in the VPG**.msg** user files located in the work directory:

   > CHECKING xxxx NODES FOR DUPLICATE NODES
   > NO DUPLICATE NODE FOUND

CHANGE ID

This function changes the numbers of the existing nodes. The parts do not need to be on or current during this procedure.

1. VPG prompts:

   > SELECT NODE FOR NEW NODE NUMBER

   • The user can select the node nearest the cursor by pressing the left mouse button instead of typing the letter “N” or entering the keyboard option.

2. After selecting a node, the user is prompted for a new node number:

   > ENTER NEW NODE NO. FOR XXXX OR [CR] FOR UNCHANGE

   • Any unused positive integer is a valid entry. If the entered number is already being used, the following prompt will appear.

   > NODE NUMBER xxxx ALREADY EXISTS, REQUEST DENIED
   > ENTER NEW NODE NO. FOR XXXX OR [CR] FOR UNCHANGE

COINCIDENT CHECK

This option checks for coincident nodes that may be in the displayed or other part(s) of the model. Coincident nodes are defined as having distance between each other that is less than or equal to a specified tolerance. The VPG default tolerance, which is the lowest tolerance VPG allows, is 0.01 units. The user may increase the default tolerance as necessary.

The COINCIDENT NODE CHECK merges two nodes in a coincident node group into one node and retains the lower node number (i.e., VPG combines two nodes that share the same location and renumbers the new node with the lower grid number). VPG designates the higher grid number as "unreferenced."

1. VPG prompts:
   > ENTER TOL (DEFAULT=0.01)
   - Any positive real number is a valid entry.

2. After entering a tolerance, VPG prompts the user to select the nodes to be checked:
   > SELECT OPTION FOR COINCIDENT NODE CHECK
   ? ALL NODES IN MODEL
   DISPLAYED NODES
   SELECTED NODE
   EXIT
   - Exits the user to the NODE OPTIONS menu without executing COINCIDENT NODE.
   - After selecting one of the previous options, or if no coincident nodes are found, the user returns to the NODE OPTIONS menu.
   - If coincident nodes are found, VPG prompts:
     > XXXX COINCIDENT NODES FOUND
     > MERGE COINCIDENT NODES (Y/N)
     - YES merges the coincident nodes.
     - NO returns the user to the NODE OPTIONS menu.
   - Coincident nodes become unreferenced nodes and are represented by asterisks (*).
   - Some or all of the unreferenced nodes may be deleted using the DELETE UNREFERENCED NODE option.
   - VPG will not merge nodes belonging to the same element (i.e. zero length elements with unique node numbers will not be merged, avoiding the degeneration the element).

COPY

This function generates a new set of unreferenced nodes at a user-specified location from either referenced or unreferenced nodes.

1. VPG prompts:
   > SELECT NODES
   - Once the nodes have been selected, DONE or EXIT concludes this step.

2. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

3. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (YES/NO)
   - If YES, the user goes to step 4. If NO, the user returns to step 2.

4. If the user selects YES, VPG prompts for the number of copies and the type of transformation:
   > NUMBER OF COPIES (positive integer)
   > MOVE OR ROTATE? (M/R)
   ? MOVE
      > ENTER U, V, W INCREMENTS
      - e.g. 0,0,1000. This entry will copy the selected node(s) 1000 units along the W-axis.

   ROTATE
      > ENTER ANGLE INCREMENT
      - Enter the angle increment (right hand rule) for local W or global Z.

   ABORT

COMPACT NODE ID

This function allows the user to DELETE either ALL or SELECTED unreferenced nodes in the database and RENUMBER the remaining nodes.

1. VPG prompts:
   > DELETE ALL OR SELECT FREE NODES? (A/S/E)
   ? ALL FREE NODES
   - Deletes all unreferenced nodes and continues with step 2.

   SELECT FREE NODES
   - The user may now select individual or grouped nodes.
   - After making the node selection, the user exits this section by pressing DONE or EXIT.
   - If any of the selected nodes are referenced nodes, the following message will echo:

   > CANNOT DELETE NODE xxxx, IT IS NOT UNREF.
   - EXIT will delete any unreferenced nodes.

2. VPG prompts:
   > ENTER STARTING NODE NUMBER
   - Users enter the number at which they want the existing nodes to br renumbered from.

CREATE (NODE)

This function generates UNREFERENCED NODES on points or at a global location.

1. VPG prompts:
   > DEFINE NODE LOCATIONS FROM POINTS

2. The user may now select node locations by:
   KEY IN X, Y, Z
   POINT
   UNDO LAST
   - By selecting DONE, the user will conclude the CREATE NODE option and return to the NODE OPTIONS menu.

DELETE UNREF. NODES

This function allows the user to DELETE either ALL or SELECTED unreferenced nodes in a database. The unreferenced nodes are labeled with asterisks (*).

1. VPG prompts:
   > DELETE ALL OR SELECT FREE NODES? (A/SF/E)
   ? ALL FREE NODES
   - Deletes all unreferenced nodes.
     SELECT FREE NODES
   - The user may now select individual or grouped nodes.
   - Select DONE or EXIT after selecting nodes.
   - If any of the selected nodes are referenced nodes, VPG displays the message:
     > CANNOT DELETE NODE xxxx, IT IS NOT UNREF.
   - EXIT will delete any unreferenced nodes. When unreferenced nodes are deleted, VPG will echo the following message:
     > xxxx UNREFERENCED NODES DELETED

DISTANCE

This function calculates the distance between two points, two nodes, or a node and point.

1. VPG prompts:
   > SELECT TWO POINTS/NODES
   - The overall distance and the global DX, DY, DZ components will be displayed at the
     prompt area of the screen.
   - The user enters ABORT, DONE, or EXIT.

IDENTIFY NODE/POINT

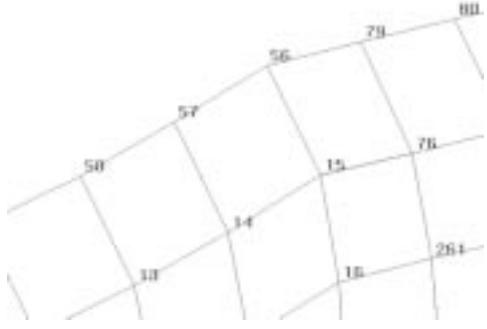
This function identifies any point/node and its corresponding global location in terms of its X, Y, Z coordinates.

1. VPG prompts:
   > PICK NODE/POINT
   - If ALL NODES is selected, VPG displays the appropriate numbers at the node locations.
   - If NODES is selected, VPG displays the number of the node nearest the cursor location. The following message will echo on the command line:
     > NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - If POINT is selected, VPG displays the number of the point nearest the cursor location. The following message will echo on the command line:
     > POINT xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - DONE or EXIT completes the command sequence and returns the user to the NODE OPTIONS menu.

note: To clear the label display, select CLEAR from the VPG DISPLAY PARAMETER OPTIONS module.

LABEL

This function toggles the node labels ON/OFF. When the user selects LA, VPG displays the numbers of the nodes in the database.



Node labels are on.

MOVE NODE TO PT/NODE

This function moves referenced or unreferenced nodes to any location on the display screen.

1. VPG prompts:
   > SELECT NODE TO MOVE
   - UNDO LAST is a valid selection only after a node has been moved. This command returns the last node moved to its previous position.
   - If a coincident node is encountered, skip to step 3 and then to step 2.

2. After selecting a node, VPG prompts for a new location for the nodes:
   > PICK NEW LOCATION
   - The user may move to an existing poin/node by mouse or keyboard entry.
   - DONE returns the user to step 1.
   - ABORT exits this function and returns the user to the NODE OPTIONS menu.

3. If multiple or coincident nodes are found near the cursor, VPG prompts:
   > COINCIDENT NODES FOUND
   > SELECT ELEMENT FOR NODE
   - If an ELEMENT center is selected by the CURSOR and it is unconnected to the target node, VPG prompts:
     > SELECTED NODE IS NOT CONNECTED TO DUPL. NODE
     > SELECT ELEMENT FOR NODE
   - After this command is completed, the user returns to step 1.

PART CONNECTION

This function determines the names and identification numbers (PID) of all of the parts that share the selected node. The message is displayed in the VPG prompt area of the screen and the nodes are highlighted with a small white circle.

1. VPG prompts:
   > SELECT NODE TO SHOW PART CONNECTION
   ? DONE
   KEY IN NODE NUMBER
   NODE
   - Once a node is selected, the part connection is listed and the user returns to step 1.
   - If the user selects a node number by KEYBOARD ENTRY that does not exist in the database, VPG prompts:
     > NODE xxxx IS NOT IN DATABASE
   - If the user selects a node number by KEYBOARD ENTRY that is not connected to a part, VPG prompts:
     > NODE xxxx IS NOT CONNECTED TO ANY PART

PROJECT

This function projects a node or group of nodes onto a plane, surface(s), or mesh. The nodes may be referenced or unreferenced (referenced would then project the finite element mesh).

1. VPG prompts:
   > SELECT PROJECT OPTION
   ? EXIT
   F.E. MESH (go to step 2)
   PLANE IN LOCAL UV (go to step 3)
   SURFACE (go to step 5)

2. Option (F):
   > SELECT PLATE ELEMENTS FOR TARGET SURFACE
   > SELECT ELEMENTS

Continued on next page.

- ABORT exits without action.
- DONE completes the operation and prompts the user to step 7.

3. Option (P)
   > P - PLANE IN LOCAL UV
   > DEFINE PROJECTION (UV) PLANE
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system to define the UV plane. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

4. VPG prompts:
   > PROJECT NODES ALONG W-AXIS? (Y/N)
   - YES go to step 9
   - NO go to step 7

5. Option (S)
   > ON SURFACE

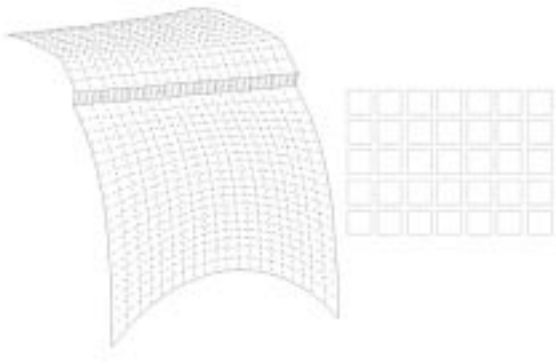
6. VPG prompts:
   > SELECT SURFACE
   - ABORT exits without action. DONE completes the operation and continues to step 7.

7. VPG Prompts:
   > **PICK NODE/PT FOR ORIGIN**
   - VPG prompts the user to create a local coordinate system to define the projection vector. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

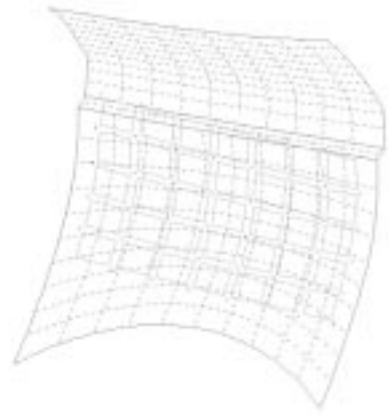
8. VPG displays the desired coordinate system and prompts:
   > ACCEPT (Y/N/A)

9. If the user selects YES, VPG prompts:
   > SELECT NODE FOR PROJECTION
   - ABORT exits without action. DONE or EXIT completes the operation and exits the PROJECT NODE menu.

Continued on next page.



Elements to be projected onto the curved surface using PROJECT NODE.



Curved surface with projection of elements.

RENUMBER NODES

Renumbers all nodes in the existing database. There is also an option of creating a summary table report of the node ranges between each part. The user may renumber nodes by sequence or parts.

1. VPG prompts:
   > OUTPUT NODE NUMBER SUMMARY TABLES (Y/N)?
   ? **YES** (see step 2)
   NO (skip to step 4)

2. If the user selects YES, VPG writes directly to the file **vpg.ntb**

3. If the file name has already been used, VPG prompts:
   > FILE ALREADY EXISTS, O.K. TO OVERWRITE? (Y/N)
   - If YES, the file will be overwritten. If NO, VPG returns to step 2 and prompts for a file name again.

Continued on next page.

4. VPG prompts for a starting node number after selecting the file name:

   > ASSIGN STARTING NODE NO. FOR EACH PART? (Y/N)

   ? YES

   > ENTER STARTING NODE NO. FOR PART " xxxx"

   • Once a number has been entered the prompt will read:

   > NODE x TO xx ASSIGNED, NEXT NODE NO.: xxxx

   > ENTER STARTING NODE NUMBER FOR PART " xxxx "

   • These prompts will continue for every part in the database.

   • If the user enters a STARTING NUMBER that already exists, VPG gives the message that the location has already been assigned and returns the user to the beginning of this step.

   > RENUMBER SUMMARY IS WRITTEN ON FILE *.ntb

   • Zero (0) will assign (MAX NODE NO. + 1).

   NO

   > ENTER STARTING NODE NUMBER, -1 TO EXIT

   • Any positive integer or zero will default to one (1).

   > RENUMBER SUMMARY IS WRITTEN ON FILE *.ntb

5. VPG prompts for the increment between node numbers:

   > ENTER NODE NUMBER INC. BETWEEN PARTS:

   > 0 - DEFAULT TO 1000, -1 NO GAP BETWEEN PARTS

   • Any integer may be entered.

SCALE

This section applies a scale factor to any or all X, Y, and Z components for some or all nodes in a global or local direction. The scale factor may be positive or negative (change the sign on any or all of the X, Y, Z components). The parts will be magnified if the scale factor is greater than 1 and reduced if the scale factor is less than 1. The user may reflect and/or enlarge or reduce a part without retaining a copy of the original grids/elements.

Continued on next page.

1. VPG prompts:
   > SELECT NODES
   - Once the user selects the desired nodes and clicks DONE, VPG prompts:

   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system. A thorough explanation of how to create a local coordinate system is covered in the *Local Coordinate System, Section 3.5*.

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ENTER X, Y, Z SCALE FACTORS
   - All real numbers are valid. ZEROS or BLANKS will default to (+1).
     EXAMPLES:
     - 1, -1, 1 will reflect the selected nodes across the XZ plane (Y-direction). All Y components will be multiplied by (-1).
     - -2, 1, 0.5 will magnify the X components by two times their size and reflect the nodes across the YZ plane (X-direction). These coordinates will not affect the Y components, but they will shrink the Z-components by half their size and reflect the nodes on the original side of the XY plane.
   - EXIT returns the user to the NODE OPTIONS menu.

TRANSFORM

This function translates or rotates the selected nodes to new locations.

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG prompts the user to create a local coordinate system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*
   - Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
     > ACCEPT? (Y/N/A)

Continued on next page.

2. If NO, the user returns to step 1. If YES, VPG prompts:

   > MOVE OR ROTATE? (M/R)

   ? MOVE

   > ENTER U, V, W INCREMENTS

   • e.g., 0,0,1000. This will move/transform the selected node(s) 1000 units along the W-axis.

   ROTATE

   > ENTER ANGLE INCREMENT (RIGHT HAND RULE) ABOUT LOCAL W OR GLOBAL Z.

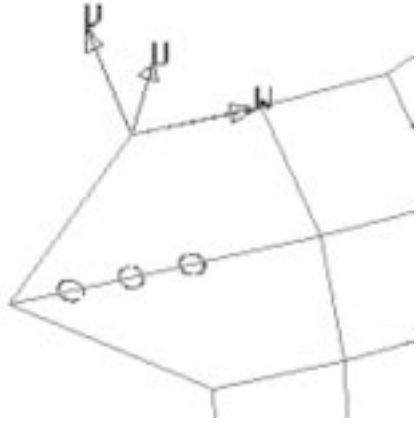
   ABORT

3. VPG prompts:

   > SELECT OPTIONS TO SELECT NODES

   • After the function for selecting the nodes and the nodes have been chosen, VPG translates them to their new location. Selecting again will repeat the translation.

4. EXIT ends this command sequence and returns the user to the NODE OPTIONS menu.



Entering AGAIN will repeat the last translation.

MODEL CHECKER

The functions in the **MODEL CHECKER** menu validate the VPG models. The values for model validation are set with defaults in VPG, but the values may be adjusted to suit the user's needs. Element orientation, size, skew, connectivity, and interior angles may be compared to the specified criteria. Elements can be checked individually or for the entire model. After a model check function is performed an interactive inquiry function allows the user to pick individual elements for the calculated quality.

MODEL CHECKER

AUTO NORMAL
AUTO ORIENT SOLID
ASPECT RATIO
BOUNDARY DISPLAY
CHECK ANGLE
CHECK OVERLAP
CHECK JACOBIAN
CHECK NORMAL
CHECK RIGID LINK
CHECK SIZE
CHECK TAPER
CHECK WARPAGE
FEATURE LINE DISPLAY
SECTION CUT
CHECK FREE SURFACE
CHECK SPRING
VIEW
EXIT

A detailed description of each function is given in the following sections.

AUTO NORMAL

This function sets the direction of the displayed plate element normals to a selected reference plate element.

1. VPG prompts:
   > SELECT PART FOR AUTO NORMAL
   ? ALL ACTIVE PARTS
   CURSOR PICK PART (DEFAULT)
   EXIT

2. User will be prompted to select a part or an element for a reference normal. The selected elements will display their current normal direction. VPG prompts:
   > IS NORMAL DIRECTION ACCEPTABLE? (Y/N)
   ? YES
   - All selected element normals will be made consistent with a referenced element normal. If normals need to be reversed for some elements VPG displays:
     > NORMALS REVERSED FOR XXXX ELEMENTS
     > PRESS LEFT MOUSE BUTTON TO CONTINUE
     > NORMALS REVERSED

   NO
   - All displayed element normals will be reversed according to the referenced element normal.

note: Selecting one part at a time is recommended. Parts that branch off or that are separated do not always get adjusted properly.

AUTO ORIENT SOLID

This function adjusts the display of the solid elements to a positive volume.

1. **AUTO ORIENT SOLID**
   - When the user selects AUTO ORIENT SOLID, no selection options are provided. The program automatically adjusts the display of solid element normals to a positive volume.
   - If elements are reversed, the following message will be returned:
     > ORIENTATION REVERSED FOR xxxx ELEMENTS

Continued on next page.

The elements that are reversed will highlight as they are checked off. The user is returned to the MODEL CHECKER menu.

- If no elements are reversed, the following message will be returned:

  > ALL SOLID ELEMENTS PASS CHECK

  No changes occur and the user is returned to the MODEL CHECKER menu.

note: The user may reverse the element normals by executing the REVERSE NORMAL command in the ELEMENT OPTIONS menu.

ASPECT RATIO

This function allows the user to check the aspect ratio of the displayed plate and solid elements. The aspect ratio is the ratio of the longest side to the shortest side of an element. In VPG, the default aspect ratio is 8.0 to 1.0, but the user may adjust it as necessary.

1. VPG prompts:

   > ENTER ASPECT RATIO (DEFAULT = 8.0)

   - Users may accept the default value of 8.0 by pressing the [Enter] key, or they may enter any real number that is a valid value for the aspect ratio.
   - VPG runs a check on the values for the aspect ratio. Any elements that exceed the desired aspect ratio are highlighted. The messages echoed to the command line are:

     > xxxx ELEMENTS FAILED CHECK or

     > ALL ELEMENTS PASS CHECK

2. The following message will be displayed for the failed elements:

   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)

   ? YES

   > ENTER NAME OF NEW PART

   - The failed elements will be removed from their original parts and added to the NEW PART.
   - The new part will be ON and CURRENT.

   NO

   - VPG prompts the user to check individual element for aspect ratio via cursor pick. Select EXIT to exit the function.

BOUNDARY DISPLAY

This function checks the continuity of the free edges in a group of elements. All free edges will be highlighted with a boundary display.

1. VPG prompts:
   > CHECK SINGLE OR MULTIPLE SURFACE? (S/M)
   SINGLE SURFACE
   - Checks the continuity of the displayed elements and highlights all of the single free edges of the displayed model.

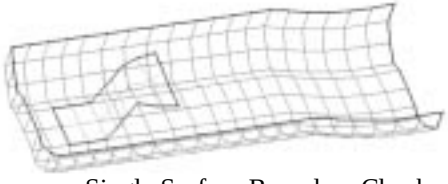
   MULTIPLE SURFACE
   - Checks the continuity of the displayed elements. It also highlights the non-intersecting free edges of the multiple surfaces unless the intersection of the multiple surfaces are not connected properly.

2. VPG prompts:
   > GENERATE PLOTEL ELEMENTS FOR FREE EDGES? (Y/N)
   ? YES
   NO (returns user to MODEL CHECKER menu)

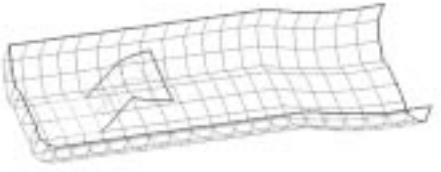
3. If the user selects YES, VPG prompts:
   > INCLUDE PLOTEL ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   NO

4. If the user includes the elements in a new part, VPG prompts:
   > ENTER NAME OF NEW PART
   - The user enters the name of the new part. After entering the name of the part, VPG prompts the user to enter a material and property for the new part. When finished, VPG will both add plotels to the part and make the part current.
   - The boundary will remain highlighted until the user selects CLEAR.

note: VPG will display the boundary of the outer most edges of a solid-element structure. Users should also use the CHECK FREE SURFACE command for solid-element structures that is described in this section.



Single Surface Boundary Check



Multiple Surfaces Boundary Check.

CHECK ANGLE

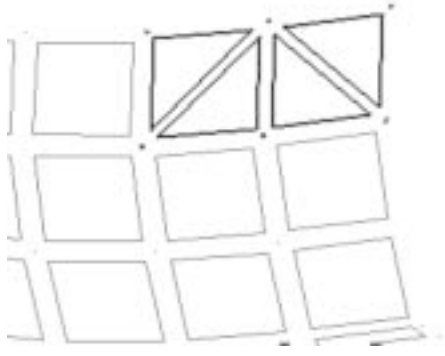
This function checks the minimum values of the interior angles for shell and solid elements. VPG checks and highlights any interior angles that are less than the default values. Users have the option to adjust the default values as necessary.

1. VPG prompts:
   > ENTER INTERIOR ANGLE (DEFAULT = 30 DEGREES)
   - If the elements pass the interior angle check, VPG echoes the following message and returns the user to the MODEL CHECKER menu:
     > ALL ELEMENTS PASS CHECK!
   - If any of the elements fail the model check, VPG displays the following message with the option to create a new part.
     > xxxx ELEMENTS FAILED CHECK

2. If the elements fail the model check, VPG prompts the user to place these elements into a new part:
   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   > ENTER NAME OF NEW PART
   - The failed elements will be removed from their original part and added to the NEW PART. The new part will be ON and CURRENT.

   NO
   - VPG prompts the user to check individual elements for interior angle via cursor pick. Select EXIT to exit this function.

The highlighted elements have failedthe interior angle check.



CHECK OVERLAP

This function checks the elements for duplicate nodal connectivity. All element types are checked.

1. **CHECK OVERLAP ELEM**
   - If no overlapping elements are found in the displayed elements, the following message is echoed and the user is returned to the **MODEL CHECKER** menu:
     > NO OVERLAP ELEMENT IN DISPLAYED ELEMENTS
   - If duplicate elements are found in the displayed elements, the following message will be echoed:
     > xxxx OVERLAP ELEMENTS FOUND
     and VPG will continue with the option of adding these elements to a new part.

2. VPG prompts:
   > INCLUDE DUPLICATE ELEMENTS IN A NEW PART? (Y/N)
   ? YES
   > ENTER NAME OF NEW PART
   - The duplicate elements will be removed from their original parts and added to the NEW PART.
   - The user will be prompted to enter a new material and property for the part.
   - The new part will be ON and CURRENT.

   NO
   - The user will exit to the MODEL CHECKER menu.

CHECK JACOBIAN

This function allows the user to check for element distortion, against the element's ideal shape, which is given a jacobian value of 1.0. The more an element is warped the closer it approaches a zero jacobian value. Elements that fail the jacobian check are highlighted and the user has the option of putting them in a separate part.

Continued on next page.

1. VPG prompts:
   > ENTER THE CRITERION FOR JACOBIAN CHECK (DEFAULT .7)

2. VPG prompts:
   > XXXX ELEMENTS FAILED CHECK
   > INCLUDE FAILED ELEMENTS IN NEW PART? (Y/N)

3. If yes, VPG prompts the user to select a name, material, and property.

4. If no, VPG prompts the user to check individual elements for JAcobian value via cursor pick. Select EXIT to exit the function.

CHECK NORMAL

This function draws a white boundary line between the opposing normals of the displayed plate elements.

1. **CHECK PLATE NORMAL**
   - VPG will highlight the boundary between the elements that have opposing normal orientations. Afterwards VPG echoes the message:
     > PLATE NORMAL CHECK COMPLETED
   - The boundary will highlight when three or more plate elements connect at a common edge.
   - PLATE NORMAL in the SETUP MENU will display the element normals.

CHECK RIGID LINK

This function checks the connectivity and length of all rigid elements, RBE2 and RBAR. Rigid bodies that exceed the length criteria of the user or that are loose, zero-length, double dependent, cyclic, or redundant are highlighted.

Continued on next page.

1. VPG prompts:
   > FIND RIGID LINKS LONGER THAN CERTAIN LENGTH? (Y/N)
   ? YES
      > ENTER LENGTH CRITERIA
      - Any real number greater than zero.

   NO

2. The user has the option to place failed elements into new parts. If the user opts to place the failed elements into a new part, VPG will generate the following part names:

   r.xlong
   - Checks for RBE2s that fail length check.

   r.duplic
   - Checks for degenerate RBE2s.

   r.cyclic
   - Checks for two RBE2s sharing the same two nodes in an independent/dependent node loop.

   r.double
   - Checks for RBE2s with double dependent nodes.

   r.loose
   - Checks for loose connections.

- The user returns to the MODEL CHECKER menu

CHECK SIZE

This function checks the minimum length of the edges of plate, solid, or beam elements. VPG highlights any elements with dimensions that are shorter than the user-defined length criteria.

Continued on next page.

1. VPG prompts:

   > ENTER LENGTH CRITERIA

   - A valid response is any real number greater than zero.
   - If the length of the of the edges of the elements is greater than the user-defined length criteria, then VPG echoes the message:

     > ALL ELEMENTS PASS CHECK!

   - If the length of the edges of the elements is less than the user-defined length, then VPG echoes one of the following messages:

     > ELEMENT xxxx FAILED CHECK or

     > xxxx ELEMENTS FAILED CHECK

2. If failed elements exist, then VPG prompts the user to place them into a part:

   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)

   ? YES

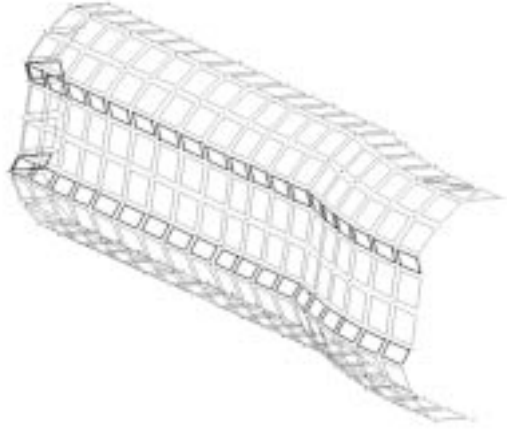
   > ENTER NAME OF NEW PART

   - The failed elements are removed from the original parts and added to the NEW PART.
   - The new part is ON and CURRENT.
   - The user may be prompted to enter the material and element properties that are undefined in the part.

   N - NO

   > SELECT ELEMENT FOR ELEMENT SIZE

   - The user can cursor check individual elements for their size.
   - DONE or EXIT returns the user to the MODEL CHECKER menu.



Highlighted elements failed the element size check.

CHECK TAPER

This function allows the user to check the length to width ratio of elements.

1. VPG prompts:
   > ENTER THE CRITERION FOR TAPER (DEFAULT 0.5)

2. VPG prompts:
   > XXXX ELEMENTS FAILED CHECK
   > INCLUDE ELEMENTS IN A NEW PART? (Y/N)
   - If yes, VPG prompts for a name, material and property for the new part.

3. VPG allows the user to check individual elements for taper value via cursor pick.

CHECK WARPAGE

This function highlights any warpage in the plate or solid elements. The user has the option to split the plate elements into triangular elements. All elements that fail may be added to a new part.

1. VPG prompts:
   > ENTER CRITERIA (5 DEG)
   - Any real number is a valid entry.
   - If the warpage of the elements is less than the entered criteria, then:
     > ALL ELEMENTS PASS CHECK!
     And the user is returned to the MODEL CHECKER menu.
     > ELEMENTS xxxx FAILED CHECK
     > xxxx ELEMENTS FAILED CHECK
   - Step 3 is not applicable for solid elements, skip to step 4 to enter the name of the new part.

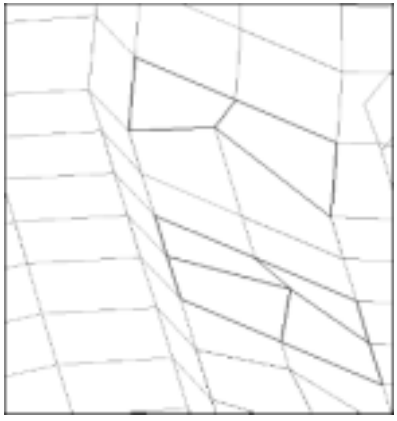
2. The user may replace warped elements:
   > REPLACE WARPED QUADS WITH TRI. ELEMENTS? (Y/N)
   ? YES
   NO (see step 4)
   - Y will replace any warped quadrilateral elements with triangular elements.

Continued on next page.

3. VPG prompts:

   > SELECT ELEMENT FOR WARPAGE

   - At this point the user can cursor pick individual elements for degree of warpage.
   - DONE or EXIT returns the user to the MODEL CHECKER menu.



An example of warped elements.

4. If some of the elements fail, and no is entered, VPG prompts the user to place them into a new part:

   > INCLUDE FAILED ELEMENTS IN A NEW PART? (Y/N)

   ? YES

   If the elements are added to a new part, VPG asks for a part name:

   > ENTER NAME OF NEW PART

   - Enter a name for the new part. The plates and/or solids will be added to this part and the part will become current. The user will exit to the MODEL CHECKER menu.

   NO

   > SELECT ELEMENT FOR WARPAGE

   - At this point the user can cursor pick individual elements for degree of warpage.
   - DONE or EXIT returns the user to the MODEL CHECKER menu. The warped solids, quadrilateral elements, or newly created triangular elements will remain in their original parts.

FEATURE LINE DISPLAY

This function allows the user to check the angle between plate or solid elements based on a user specified angle criteria. The user can also automatically generate plotel elements at these locations to display the structures main features.

1. VPG prompts:
   > ENTER FEATURE LINE ANGLE CRITERION (DEFAULT: 20DEG)

2. VPG prompts:
   > CHECKING FEATURE LINES
   > GENERATE PLOTEL ELEMENTS FOR FEATURE LINES? (Y/N)
   - NO returns the user to the model checker menu.
   - YES prompts the following:

3. VPG prompts:
   > INCLUDE PLOTEL ELEMENTS IN A NEW PART? (Y/N)
   - NO - The plotel elements will be included in the current part.
   - YES - VPG prompts for a part name.

SECTION CUT

This function cuts through the plate and solid elements at a UV-plane that the user defines and creates plotel elements at that plane.

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is now asking for a coordinate system to create section plotels. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

2. Once the desired coordinate system is acquired, it will be displayed on the screen and the prompt will read:
   > ACCEPT? (Y/N/A)
   ? YES

Continued on next page.

> ENTER DISTANCE OF SECTION PLANE FROM W = 0

- Any real value is a valid response.
- User may now input a distance along the W-axis from the selected origin of the coordinate.
- The cross section of plotels will be created at the distance W in the UV-plane.

NO

- Returns to step 1.

ABORT

3. VPG prompts:

   > INCLUDE PLOTEL ELEMENTS IN A NEW PART? (Y/N)

   ? YES

   NO

   - I f no is selected, plotel elements will be added to the current part and the user will exit to the MODEL CHECKER menu.

4. If elements are added to a new part, VPG asks for a part name:

   > ENTER NAME OF NEW PART

   - After entering the name of the part, the user will be prompted to enter a new material and property for the part.
   - The plotel elements will be included in the new part.
   - The new part will be ON and CURRENT.

CHECK FREE SURFACE

This function checks the continuity of the solid elements. A free surface is the face of a solid element that is occupied by one solid element. VPG will replot the solid elements on screen displaying only the free surfaces. Any unexpected quad or tri surface indicates potential errors in the solid element connectivity.

1. **CHECK FREE SURFACE**

   - The following five messages will appear in the command line area of the screen:

   > XXXX SURFACE FOUND
   > CHECKING FOR FREE SURFACES
   > XXXX SURFACES CHECKED
   > CONSTRUCTING FREE SURFACES
   > xxxx FREE SURFACES FOUND

Continued on next page.

- The user is now placed in a **FREE SURFACE** sub-menu.

2. **FREE SURFACE**
   - The following are brief descriptions of the options for the FREE SURFACE sub-menu:

     DISTANCE BETWEEN PT/NODE
     - This measures the distance between the points and nodes.

     IDENTIFY ELEMENT
     - This identifies the element numbers of the plate elements that represent the free surfaces in the FREE SURFACE sub-menu. Each plate element ID is identical to the solid element ID (i.e., one solid element with the ID xyz will have the plate elements that correspond with ID xyz).

     IDENTIFY NODE
     - Identifies the nodes in the solid elements and acquires the node numbers and the X, Y, Z global coordinates.

     CREATE ELEMENTS
     - Creates plate elements on the free surfaces of the displayed solids. The plate elements will reside on the screen in the current part at the time of creation.
     - In order to for the user to identify any unwanted free surfaces, VPG creates plate elements that carry element/node numbers from a parent solid element. If these plate elements are used in a model, they should be renumbered with the RENUMBER ELEMENT and RENUMBER NODE commands in the ELEMENT OPTIONS and NODE OPTIONS menus respectively.

3. EXIT FREE SURFACES returns the user to the MODEL CHECKER menu.

CHECK SPRING

This function checks spring elements for unconnected ends. If unconnected elements are found, a mass value can be entered to create mass elements at the free ends automatically.

SET MENU (LS-DYNA)

The functions in the SET MENU create sets that facilitate the organization of data specification and output control data.

The following options are available in the SET MENU when the analysis code is set to LS-DYNA:

BEAM
DISCRETE
NODE
PART
SEGMENT
SHELL
SOLID
THICK SHELL
EXIT

note: when the analysis code is set to NASTRAN see section 6.7.

Each option contains a similar submenu with the following options for manipulating the output control of their respective set groups:

ANNOTATE
CREATE
CURRENT
DELETE
ERASE EMPTY SETS
ERASE UNREF. SETS
INCLUDE (ELEMENTS/NODES/PARTS...)
LIST
REMOVE (ELEMENTS/NODES/PARTS...)
REVIEW LIST
SHOW SET
SET ON/OFF (PART)
VIEW
EXIT

A detailed description of each function is given in the following sections.

ANNOTATE

This function allows the user to add notes to a selected set.

1. VPG prompts:
   > SELECT SET FROM (type) SET LIST
   ? [List of all sets]
   CURSOR AT (element/node/part)
   EXIT

2. After selecting a set VPG prompts:
   > SET X IS SELECTED
   > ENTER CARD DESCRIPTION (70 CHAR. MAX)
   - Once the note is entered the user is returned to step 1.

CREATE

This function allows the user to generate sets with unique ID numbers. The respective element/node/part may be included in multiple sets.

1. VPG prompts:
   > ENTER NEW (type) SET NUMBER (X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.
   - If the user enters a set ID number already assigned, VPG returns the prompt:
     > (type) SET xxxx ALREADY EXISTS

2. VPG prompts:
   > SELECT (element/node/part)
   - The user may press the left most mouse button or "N" to select the element nearest to the cursor.
   - If the user selects the same element/node/part twice while creating a the set, VPG returns the prompt:
     > (type) xxxx ALREADY PICKED
   - If the user selects **DONE/EXIT**, VPG saves the set and prompts:
     > X (type) INCLUDED IN (type) SET X
   - If the user selects ABORT, VPG deletes the node set and returns the user to the SET submenu.

CURRENT

This function allows the user to select the current set from a list of previously defined sets.

1. VPG prompts:
   ```
   >   SELECT SET FROM (type) SET LIST
   ?   [List of all sets]
       ALL (type) SETS
       EXIT
   ```

2. Once the set(s) is selected, VPG prompts:
   ```
   >   SET X IS FLAGGED AS CURRENT SET
   ```
   - The selected set is current and listed in the lower right hand corner of the display window.

DELETE

This function allows the user to remove sets by ID number. Clicking on the set name also indicates where the set has been used such as in boundary conditions. (VPG is specifically configured to recognize sets for this procedure.)

1. VPG prompts:
   ```
   >   SELECT SET FROM (type) SET LIST
   ?   [List of all sets]
       ALL (type) SETS
       EXIT
   ```

2. If the user selects ALL (type) SETS in step 2, the sets are highlighted in red in the VPG MENU WINDOW.
   ```
   >   O.K. TO DELETE ALL (type) SETS? (Y/N)
   ?   YES
       NO
   ```
   - If YES, VPG issues the following message:
   ```
   >   ALL (type) SETS DELETED FROM DATABASE
   ```
   - If NO, VPG returns the user to the set submenu.

3. If the user selects an individual set in step 1, the set will be highlighted in red. When the user exits the submenu all highlighted sets will be deleted.

ERASE EMPTY SETS (NODE)

This function deletes sets with no nodes assigned to them.

ERASE UNREFERENCED SETS (NODE)

This function deletes sets that have not been assigned boundary conditions, constraint, initial conditions, loads or contact.

INCLUDE

This function adds additional elements/nodes/parts to a previously defined set.

1. VPG prompts:
   > SELECT (element/node/part) FOR (type) SET XXXX
   > SELECT (element/node/part)

2. Once the elements/nodes/parts have been selected and the user has exited the IN - INCLUDE submenu VPG displays the new additions with their set number and prompts:
   > XXXX (ELEMENTS/NODES/PARTS) IN SET XXXX

LIST

This function lists the defined sets in a pop-up window.

REMOVE

This function deletes elements/nodes/parts from a previously defined set.

1. VPG prompts:
   > SELECT (element/node/part) FROM (type) SET XXXX
   > SELECT (element/node/part)

2. Once the elements/nodes/parts have been selected and the user has exited the - REMOVE submenu, VPG removes the elements/nodes/part from their set.

REVIEW LIST

This function displays the defined sets in the MENU WINDOW. When a set is selected the elements/nodes/parts of that set are displayed in the CONTENTS OF SET window.

SHOW SET

This function allows the user to display all or some of the defined sets. VPG displays an ID number next to each element/node/part.

1. VPG prompts:
   > SELECT SET FROM (type) SET LIST
   ? [List of all sets]
   ALL (type) SETS
   EXIT
   - VPG displays all node sets and returns the user to the set sub-menu.

note: Because more than one set ID number may be assigned to each element/node/part, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific set at a time.

SET ON/OFF (PART)

This function allows the user to turn on and off parts included in a selected part set. It allows for convenient organization of parts in a complicated model and manipulation of large numbers of parts in sub-assemblage level.

SET MENU (NASTRAN)

The functions in the **SET MENU** create node and element sets that facilitate the organization of output control data.

The following three options are available in the **SET** menu when the analysis code is set to NASTRAN in the SETUP MENU:

NODE SET
ELEMENT SET
PART SET
EXIT

note: If the analysis code is set to LS-DYNA see section 6.6.

Each option contains a similar submenu with the following options for manipulating the output control of their respective set groups:

CREATE
CURRENT
DELETE
INCLUDE (NODES/ELEMENTS/PARTS)
REMOVE (NODES/ELEMENTS/PARTS)
SHOW SET
SET ON/OFF (PART)
VIEW
EXIT

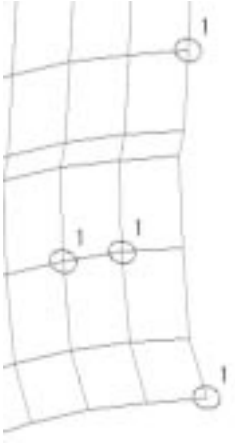
A detailed description of each function is given in the following sections.

CREATE SET (NODES)

This function allows the user to generate sets with unique node and element ID numbers. The nodes may be included in multiple sets.

1. VPG prompts:
   > ENTER NEW NODE SET NUMBER (X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.
   - If the user enters a node set ID number already assigned, VPG returns the prompt:
     > NODE SET xxxx ALREADY EXISTS

Node set 1 is created.



2. VPG prompts:
   > SELECT NODES
   - The user may press the left most mouse button or "N" to select the node nearest to the cursor.
   - If the user selects the same node twice while creating a node set, VPG returns the prompt:
     > NODE xxxx ALREADY PICKED
   - If the user enters **DONE**, VPG saves the set and prompts:
     > X NODES INCLUDED IN SET X
     > ENTER NEW NODE SET NUMBER (X) OR E TO EXIT
   - If the user selects EXIT, VPG saves the set and returns the user to step 2 .
   - If the user selects ABORT, VPG deletes the node set and returns the user to the NODE SET submenu.

CURRENT SET (NODES)

This function lists the defined node sets in a pop-up window. The user can then select the current set from those listed.

1. VPG prompts:
   ```
   >   SELECT SET FROM NODE SET LIST
   ?   [List of all node sets]
       CURSOR AT NODE
       EXIT
   ```

2. Once the set(s) is selected, VPG prompts:
   ```
   >   SET X IS SELECTED
   ```

DELETE SET (NODES)

This function allows the user to remove node sets by ID number. (VPG is specifically configured to recognize node sets for this procedure. For deleting element sets, refer to the command on page 9 of this section.)

1. VPG prompts:
   ```
   >   SELECT SET FROM NODE SET LIST
   ?   [List of all node sets]
       CURSOR AT NODE
       ALL NODE SETS
       UNDO
       EXIT
   ```

2. If the user selects **ALL NODE SETS** in step 2, VPG prompts:
   ```
   >   O.K. TO DELETE ALL NODE SETS? (Y/N)
   ?   YES
       NO
   ```
   - If YES, VPG issues the following message:
   ```
   >   ALL NODE SETS DELETED FROM DATABASE
   ```
   - If NO, VPG returns the user to the NODE SET submenu.

3. If the user selects an individual node set in step 1, VPG prompts:
   ```
   >   SET X IS SELECTED
   ```

Continued on next page.

```
>   O.K. TO DELETE ALL SETS? (Y/N)
?   YES
    NO
```

- If YES, VPG deletes the node set and returns the user to step 1.
- If NO, VPG saves the node set and returns the user to step 1.

4. EXIT returns the user to the NODE SET submenu.

INCLUDE NODES

This function adds additional nodes to the current node set.

1. VPG prompts:

```
>   SELECT NODES FOR NODE SET XXXX
>   SELECT NODES
```

2. Once the nodes have been selected and the user has exited the INCLUDE submenu VPG displays the nodes with their node set number

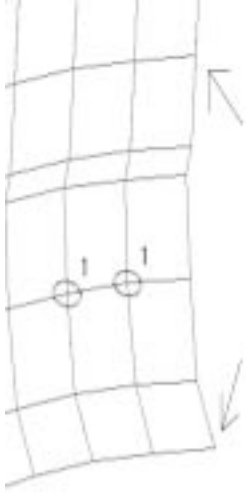
Nodes included in node set 1.

REMOVE NODES

This function deletes selected nodes from the current node set.

1. VPG prompts:
   > SELECT NODES FOR NODE SET XXXX
   > SELECT NODES

2. Once the nodes have been selected and the user has exited the REMOVE submenu VPG removes the nodes from their node set.



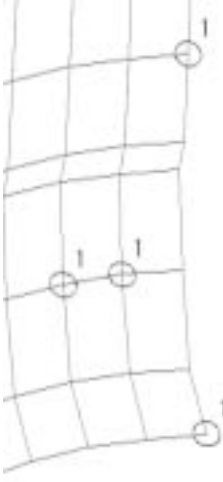
Nodes removed from node set 1.

SHOW SET (NODES)

This function allows the user to display all or some of the defined node sets. VPG displays an ID number next to each node.

Continued on next page.

1. VPG prompts:
   > SELECT SET FROM NODE SET LIST
   ? [List of all node sets]
   CURSOR AT NODE
   SELECTED NODE SETS
   UNDO
   EXIT
   - VPG displays all node sets and returns the user to the NODE SET sub-menu.



Node set 1 is displayed.

note: Because more than one node set ID number may be assigned to each node, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific node set at a time.

CREATE SET (ELEMENTS)

This function allows the user to generate sets with unique element and node ID numbers. The elements may be included in multiple sets.

1. VPG prompts:
   > ENTER NEW ELEMENT SET NUMBER(X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.

Continued on next page.

- If the user enters an element set ID number already assigned, VPG returns the prompt:

  > NODE SET xxxx ALREADY EXISTS

2. After the user enters an element set ID number, VPG prompts:

   > SELECT ELEMENT SET TYPE
   BEAM
   SHELL
   SOLID
   DISCRETE
   THICK SHELL
   EXIT

3. VPG prompts:

   > SELECT (TYPE) ELEMENTS FOR ELEMENT SET X
   > SELECT ELEMENTS
   ? ABORT
   CURSOR LOCATION
   DONE
   EXIT
   KEY IN ELEM RANGE
   PART

   > PICK A LINE (L), ELEM (C), OR PART NAME (P) OF A PART

   - The user must select a target part:

   MULTI-POINT REGION
   REJECT LAST
   SURFACE
   ELEMENT TYPE
   DRAG WINDOW
   EXCLUDE ON/OFF

   - The user may press the left most mouse button or type a "C" to select the element nearest the cursor.
   - If the user selects the same element twice while creating a set, VPG returns the prompt:

     > ELEMENT xxxx ALREADY PICKED

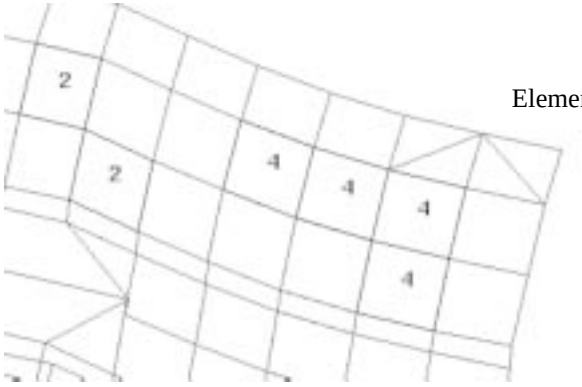
   - If the user selects DONE, VPG saves the set, returns the user to step 2 and prompts:

     > X ELEMENTS INCLUDED IN ELEM SET X

Continued on next page.

- If the user selects EXIT, VPG saves the set and returns the user to step 2 .
- If the user selects ABORT, VPG deletes the set and forwards the user to the ELEMENT SET submenu.

note: Users may not assign the same ID numbers to both node sets and element sets.



Element sets 2 and 4 are created.

CURRENT SET (ELEMENTS)

This function lists the defined element sets in a pop-up window. The user can then select the current set from those listed.

1. VPG prompts:
   > SELECT SET FROM ELEMENT SET LIST
   ? [List of all element sets]
   CURSOR AT ELEMENT
   EXIT

2. Once the set(s) is selected, VPG prompts:
   > SET X IS SELECTED

DELETE SET (ELEMENTS)

This function allows the user to remove element sets by ID number. (VPG is specifically configured to recognize element sets for this procedure. For deleting element nodes, refer to the command on page 3 of this section.)

Continued on next page.

1. VPG prompts:
   ```
   >   SELECT SET FROM ELEMENT SET LIST
   ?   [List of all element sets]
       CURSOR AT ELEMENT
       ALL ELEMENT SETS
       UNDO
       EXIT
   ```

2. If the user selects ALL ELEMENT SETS in step 1, VPG prompts:
   ```
   >   O.K. TO DELETE ALL ELEMENT SETS? (Y/N)
   ?   YES
       NO
   ```
   - If **YES**, VPG issues the following message:
     ```
     >   ALL ELEMENT SETS DELETED FROM DATABASE
     ```
   - If **NO**, VPG returns the user to the **ELEMENT SET** submenu.

3. If the user selects an individual element set in step 1, VPG prompts:
   ```
   >   O.K. TO DELETE ELEMENT SET  xxxx?
   ?   YES
       NO
   ```
   - If the user selects YES, VPG deletes the element set and returns the user to step 1.
   - If the user selects NO, VPG retains the element set and returns the user to step 1.

4. EXIT returns the user to the ELEMENT SET submenu.

INCLUDE (ELEMENTS)

This function adds additional elements to the current element set.

1. VPG prompts:
   ```
   >   SELECT (ELEMENT TYPE) ELEMENTS FOR ELEMENT SET XXXX
   >   SELECT ELEMENTS
   ```

2. When the user has finished selecting elements and exited the menu, VPG prompts:
   ```
   >   xxxx ELEMENTS SELECTED
   ```
   - The selected elements are added to the set and numbered accordingly.

REMOVE (ELEMENTS)

This function deletes elements from the current element set.

1. VPG prompts:
   > SELECT ELEMENTS FROM ELEMENT SET XXXX
   > SELECT ELEMENTS

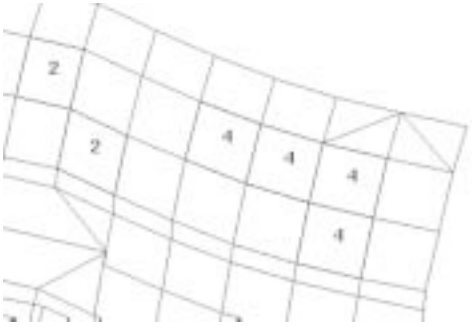
2. When the user has finished selecting elements and exited the menu, VPG prompts:
   > xxxx ELEMENTS REMOVED FROM SET XXXX

SHOW SET (ELEMENTS)

This function allows the user to display all or some of the defined element sets. VPG displays the appropriate ID numbers on the screen next to each defined element.

1. VPG prompts:
   > SELECT SET FROM ELEMENT SET LIST
   ? [List of all element sets]
   CURSOR AT ELEMENT
   ALL ELEMENT SETS
   UNDO
   EXIT
   - VPG displays all element sets and returns the user to the ELEMENT SET submenu.

Element sets 2 and 4 are displayed.



note: Because more than one ID number may be assigned to each element, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific element set at a time.

CREATE SET (PARTS)

This function allows the user to generate part sets with unique element and node ID numbers. The parts may be included in multiple sets.

1. VPG prompts:
   > ENTER NEW PART SET NUMBER(X) OR E TO EXIT
   - Any integer greater than zero (0) is valid.

2 VPG prompts:
   > PICK AN ELEMENT OR PART NAME OF A PART
   [list of all parts]
   - The user may press the left most mouse button or type a "**C**" to select the element nearest the cursor.

3. VPG prompts:
   > PART XXXX IS SELECTED
   > PICK AN ELEMENT OR PART NAME OF A PART
   [list of all parts]

4. When the user is finished and selects EXIT, VPG prompts:
   > X PARTS INCLUDED IN PART SET X
   - VPG returns to step 2.

CURRENT SET (PARTS)

This function lists the defined part sets in a pop-up window. The user can then select the current set from those listed.

1. VPG prompts:
   > SELECT SET FROM PART SET LIST
   ? [List of all part sets]
   ALL PART SETS
   EXIT

2. Once the set(s) is selected, VPG prompts:
   > SET X IS SELECTED

DELETE SET (PARTS)

This function allows the user to remove part sets by ID number. (VPG is specifically configured to recognize part sets for this procedure. For deleting part nodes, refer to the command REMOVE NODES earlier in this section.)

1. VPG prompts:
   ```
   >   SELECT SET FROM PART SET LIST
   ?   [List of all PART sets]
       CURSOR AT ELEMENT
       EXIT
   ```

2. If the user selects ALL PART SETS in step 2, VPG prompts:
   ```
   >   O.K. TO DELETE ALL PART SETS? (Y/N)
   ?   YES
       NO
   ```
   - If YES, VPG issues the following message:
     ```
     >   ALL PART SETS DELETED FROM DATABASE
     ```
   - If NO, VPG returns the user to the PART SET submenu.

3. If the user selects an individual element set in step 2, VPG prompts:
   ```
   >   O.K. TO DELETE PART SET  xxxx?
   ?   YES
       NO
   ```
   - If the user selects YES, VPG deletes the element set and returns the user to step 1.
   - If the user selects NO, VPG retains the element set and returns the user to step 1.

4. EXIT returns the user to the PART SET submenu.

INCLUDE (PARTS)

This function adds additional parts to the current part set.

Continued on next page.

1. VPG prompts:
   > SELECT PARTS FOR PART SET XXXX
   > PICK AN ELEMENT OR PART NAME OF A PART

2. Once the user has selected a part, VPG prompts:
   > PART XXXX IS SELECTED
   - The user is returned to step 1 until EXIT is selected.

REMOVE (PARTS)

This function deletes parts from a previously defined part set.

1. VPG prompts:
   > SELECT PARTS FOR PART SET XXXX
   > PICK AN ELEMENT OR PART NAME OF A PART

2. When the user has finished selecting elements and exited the menu VPG removes the selected parts from the set.

SHOW SET (PARTS)

This function allows the user to display all or some of the defined part sets. VPG displays the appropriate ID numbers on the screen next to each defined part.

1. VPG prompts:
   > SELECT SET FROM PART SET LIST
   ? [List of all part sets]
   CURSOR AT ELEMENT
   EXIT
   - VPG displays the selected part sets and returns the user to the PART SET submenu.

note: Because more than one ID number may be assigned to each element, viewing all of the ID numbers at once may be confusing. The user may clarify the screen by displaying one specific element set at a time.

SET ON/OFF (PART)

This function allows the user to turn on and off parts included in a selected part set. It allows for convenient organization of parts in a complicated model and manipulation of large numbers of parts in sub-assemblage level.

BOUNDARY CONDITIONS (LS-DYNA)

The functions of the BOUNDARY CONDITIONS MENU create and verify constraints and loads on finite element models. The functions in the BOUNDARY CONDITIONS MENU are organized as follows:

B.C., CONSTRAINTS, LOADS

BOUNDARY CARDS
CONSTRAINT CARDS
INIT. COND. CARDS
LOAD CARDS
EXIT

A detailed description of each function is given in the following sections.

note: If the selected analysis code is NASTRAN a separate menu with different options will appear. See Section 6.9BOUNDARY CONDITIONS (LS-DYNA).

BOUNDARY CARDS

The BOUNDARY CARDS functions create and verify constraints and loads on finite element models. The functions in the BOUNDARY CARDS MENU are organized as follows:

1. VPG prompts:

```
>   SELECT OPTION
    CONVECTION
    CYCLIC
    FLUX
    NON-REFLECTING
    NON-REFLECTING2D
    PRESCRIBED MOTION
    PRESSURE OUTFLOW
    RADIATION
    SLIDING PLANE
    SPC
    SYMMETRY FAILURE
    TEMEPRATURE
    EXIT
```

note: All of these options share this standard submenu. The submenu yields varying ways to access and create definition cards for the above BOUNDARY CARDS.

> CREATE

- Defines a new card.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

-BOUNDARY CARDS

CONVECTION

This function defines convection boundary conditions for a thermal or coupled thermal/ structural analysis.

Continued on next page.

Refer to the LS-DYNA User's Manual (Section 3.1) for further description.

-BOUNDARY CARDS
CYCLIC

These boundary conditions can be used to model a segment of an object that has rotational symmetry such as an impeller. The segment boundaries may be curved or planar.

Refer to the LS-DYNA User's Manual (Section 3.4) for further description.

-BOUNDARY CARDS
FLUX

This function defines flux boundary conditions for a thermal or coupled thermal/structural analysis.

Refer to the LS-DYNA User's Manual (Section 3.6) for further description.

-BOUNDARY CARDS
NON-REFLECTING

This function defines a non-reflecting boundary. The option applies to continuum domains modeled with solid elements, as indefinite domains are usually not modeled. For geomechanical problems this option is important for limiting the size of the models.

Refer to the LS-DYNA User's Manual (Section 3.9) for further description.

-BOUNDARY CARDS
NON-REFLECTING2D

Refer to the LS-DYNA User's Manual (Section 3.11) for further description.

This function defines a non-reflecting boundary. The option applies to continuum domains modeled with two dimensional solid elements in the xy plane, as indefinite domains are usually not modelled. For geomechanical problems this option is important for limiting the size of the models.

-BOUNDARY CARDS
PRESCRIBED MOTION

This function defines an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. Also, velocities and displacements can be imposed on rigid bodies. If the local option is active, the motion is prescribed with respect to the local coordinate system for the rigid body.

Refer to the LS-DYNA User's Manual (Section 4) for further description.

-BOUNDARY CARDS
PRESSURE OUTFLOW

This function defines pressure outflow boundary conditions. These boundary conditions are attached to solid elements using the eulerian ambient formulation and defined to be pressure outflow ambient elements.

Refer to the LS-DYNA User's Manual (Section 3.17) for further description.

-BOUNDARY CARDS
RADIATION

This function defines radiation boundary conditions for a thermal or coupled thermal/ structural analysis.

The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 3.19) for further description.

-BOUNDARY CARDS
SLIDING PLANE

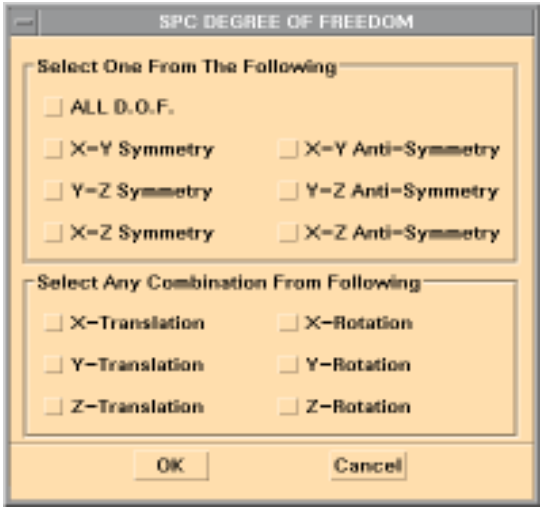
This function defines a sliding symmetry plane. The option applies to continuum domains modeled with solid elements.

Refer to the LS-DYNA User's Manual (Section 3.21) for further description.

-BOUNDARY CARDS

SPC

This function defines nodal single point constraints. After selecting CREATE, FEMB displays the following dialogue box. Once the DOF is determined in the dialogue box, FEMB prompts the user to select the nodes. DONE finishes the selection and displays the nodes with their SPC set number.



The options in the top and botom of the SPC DEGREE OF FREEDOM box represent alternate methods for defining the degree of freedom. Only the top or the bottom can be selected; not both.

X-Y SYMMETRY
- Assigns symmetric boundary conditions to the XY plane.

Y-Z SYMMETRY
- Assigns symmetric boundary conditions to the YZ plane.

X-Z SYMMETRY
- Assigns symmetric boundary conditions to the XZ plane.

X-Y ANTI-SYMMETRY
- Assigns asymmetric boundary conditions to the XY plane.

Y-Z ANTI-SYMMETRY
- Assigns asymmetric boundary conditions to the YZ plane.

X-Z ANTI-SYMMETRY
- Assigns asymmetric boundary conditions to the XZ plane.

Continued on next page.

X - TRANSLATION
- X-translation will be constrained.

Y - TRANSLATION
- Y-translation will be constrained.

Z - TRANSLATION
- Z-translation will be constrained.

X - ROTATION
- X-rotation will be constrained.

Y - ROTATION
- Y-rotation will be constrained.

Z - ROTATION
- Z-rotation will be constrained.

-BOUNDARY CARDS
SYMMETRY FAILURE

This function defines a symmetry plane with a failure criterion. This option applies to continuum domains modeled with solid elements.

Refer to the LS-DYNA User's Manual (Section 3.24) for further description.

CONSTRAINT CARDS

The functions in the CONSTRAINT CARDS menu provide a means of constraining the degrees of freedom in model parts.

EXTRA NODES
GENERALIZED WELD
JOINT
JOINT STIFFNESS
LINEAR
LAGRANGE IN SOLID
NODAL RIGID BODY
NODE SET
RIGID BODIES
SHELL TO SOLID
SPOTWELD
TIE-BREAK
TIED NODES W/FAILURE
EXIT

note: All of these options share this standard submenu. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE

- Defines a new card.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

-CONSTRAINT CARDS
EXTRA NODES

This function defines extra nodes for a rigid body.

1. When CREATE is selected, VPG prompts:

   > SELECT OPTION
   BY NODE
   BY SET

   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 4.2) for further description.

-CONSTRAINT CARDS
GENERALIZED WELD

This function defines spot and fillet welds. Coincident nodes are permitted if the local coordinate ID is defined. For the spot weld a local coordinate ID is not required if the nodes are offset. Failures can include both the plastic and brittle failures. These can be used either independently of together. Failure occurs when either criteria is met. The welds may undergo large rotations since the equations of rigid body mechanics are used to update their motion.

1. When CREATE is selected, VPG prompts:

   ```
   >  SELECT OPTION
      BUTTWELD
      COMBINED WELD
      CROSS FILLET WELD
      FILLET WELD
      SPOTWELD
      EXIT
   ```

   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 4.4) for further description.

-CONSTRAINT CARDS

JOINT STIFFNESS

This function defines joint stiffness for joints defined by the previous option.

1. When CREATE is selected, VPG prompts:

   ```
   >  SELECT OPTION
      FLEXION-TORSION
      GENERALIZED
   ```

   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 4.23) for further description.

Continued on next page.

-CONSTRAINT CARDS
LINEAR

This function defines linear constraint equations between displacements/rotations, which can be defined in local/global coordinate systems.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.35) for further description.

-CONSTRAINT CARDS
LAGRANGE IN SOLID

This function couples Lagrangian mesh shells, solids or beams to the material points of a Eulerain flow (master). This option may also be used to model rebar in concrete or tire cords in rubber. The slave part or slave part set is coupled to the master part or master part set.

1 When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.34) for further description.

-CONSTRAINT CARDS
NODAL RIGID BODY

This function defines a nodal rigid body. This is a rigid body which consists of the defined nodes. If the INERTIA option is not used, then the inertia tensor is computed form the nodal masses. Arbitrary motion of this rigid body is allowed. If the INERTIA option is used, constant translational and rotational velocities can be defined in a global or local coordinate system.

Continued on next page.

1 When CREATE is selected, VPG prompts:

```
> SELECT OPTION
  BY NODE
  BY SET
```

- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 4.39) for further description.

-CONSTRAINT CARDS
NODE SET

This function defines nodal constraint sets for translational motion in global coordinates. No rotational coupling.

1 When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.44) for further description.

-CONSTRAINT CARDS
RIGID BODIES

This function merges two rigid bodies. One rigid body, the slave rigid body, is merged to the other one called the master rigid body.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.46) for further description.

-CONSTRAINT CARDS
SHELL TO SOLID

This function defines a tie between a shell edge and solid elements. Nodal rigid bodies can perform the same function and may also be used.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.53) for further description.

-CONSTRAINT CARDS
SPOTWELD

This function defines massless spot welds between noncontiguous nodal pairs. A rigid beam is assumed between the nodal pairs; thus, nodal rotations and displacements are coupled. The spot welds must be connected to nodes having rotary inertias, i.e., beams or shell. Note that shell elements do not have rotary stiffness in the normal direction and, therefore, this component will not be transmitted. Spotwelded nodes must not have the same coordinates.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.55) for further description.

-CONSTRAINT CARDS
TIE-BREAK

This function defines a tied shell edge interface that can release locally as a function of plastic strain of the shells surrounding the interface nodes. A rather ductile failure is achieved.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.58) for further description.

-CONSTRAINT CARDS
TIED NODES W/FAIL

This function defines a tied node set with the failure based on plastic strain. The nodes must be coincident.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 4.60) for further description.

INIT. COND. CARDS

Initial condition cards provide a way of initializing velocities and detonation points.

1. VPG prompts:

```
>   SELECT OPTION
    DETONATION
    MOMENTUM
    STRESS
    TEMPERATURE
    VELOCITY
    VOID
    EXIT
```

note: All of these options share this standard submenu. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE

- Defines a new card.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

-INIT. COND. CARDS
DETONATION

This card defines points to initiate the location of high explosive detonations in part ID's which use material type 8.

Refer to the LS-DYNA User's Manual (Section 15.2) for further description.

-INIT. COND. CARDS
MOMENTUM

This card defines initial momentum to be deposited in solid elements.

Refer to the LS-DYNA User's Manual (Section 15.6) for further description.

-INIT. COND. CARDS
STRESS

This card defines initial stress and plastic strains in the Hughes-Liu beam elements.

Refer to the LS-DYNA User's Manual (Sections 15.8, 15.10, 15.12) for further description.

-INIT. COND. CARDS
TEMPERATURE

This card defines initial nodal point temperatures using nodal set ID's or node numbers. These initial temperatures are used in a thermal only analysis or a coupled thermal/ structural analysis.

Refer to the LS-DYNA User's Manual (Section 15.14) for further description.

-INIT. COND. CARDS
VELOCITY

This card defines initial nodal point translational velocities using nodal set ID's. This may also be used for sets in which some nodes have other velocities.

Refer to the LS-DYNA User's Manual (Section 15.15, 15.17, 15.18) for further description.

-INIT. COND. CARDS
VOID

This card defines initial voided part set ID's or part numbers.

Refer to the LS-DYNA User's Manual (Section 15.20) for further description.

LOAD CARDS

The LOAD CARD submenu contains cards that define applied forces.

1. VPG prompts:

```
>   SELECT OPTION
    BEAM
    BODY
    BRODE (explosive)
    DENSITY
    HEAT GENERATION
    NODE
    RIGID BODY
    SEGMENT PRESSURE
    PLATE PRESSURE
    SUPERPLASTIC FORM
    SUBSEA
    THERMAL
    EXIT
```

note: All of these options share this standard submenu. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE
- Defines a new card.

DELETE
- Deletes a selected card or cards from those displayed in a pop-up window.

LIST
- Lists the cards in the option's database via pop-up window.

MODIFY
- Similar to the CREATE command this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

-LOAD CARDS
BEAM

This card defines a distributed traction load along any local axis of beam or set of beams.

Refer to the LS-DYNA User's Manual (Section 18.2) for further description.

-LOAD CARDS
BODY

This card defines body force loads due to a prescribed base acceleration or angular velocity using global axes directions.

Refer to the LS-DYNA User's Manual (Section 18.4, 18.8) for further description.

-LOAD CARDS
BRODE

This card defines pressure loads due to explosion.

Refer to the LS-DYNA User's Manual (Section 18.11) for further description.

-LOAD CARDS
DENSITY

This card defines density versus depth for gravity loading.

Refer to the LS-DYNA User's Manual (Section 18.13) for further description.

-LOAD CARDS

HEAT GENERATION

This card defines solid elements or solid element sets with heat generation.

Refer to the LS-DYNA User's Manual (Section 18.14) for further description.

-LOAD CARDS

NODE

This card defines a concentrated nodal force to a node or a set of nodes.

Refer to the LS-DYNA User's Manual (Section 18.15) for further description.

-LOAD CARDS

RIGID BODY

This card applies a concentrated nodal force to a rigid body. The force is applied at the center of mass or a moment is applied around a global axis.

Refer to the LS-DYNA User's Manual (Section 18.18) for further description.

-LOAD CARDS

SEGMENT PRESSURE

This card defines a distributed pressure load over one triangular or quadrilateral segment defined by four nodes.

Refer to the LS-DYNA User's Manual (Section 18.20) for further description.

-LOAD CARDS

SHELL PRESSURE

This card defines a distributed pressure load over one shell element or shell element set.

Refer to the LS-DYNA User's Manual (Section 18.24) for further description.

-LOAD CARDS

SUPER PLASTIC FORMING

This card is used to perform superplastic forming analyses. This option can be applied to both solid and shell elements.

Refer to the LS-DYNA User's Manual (Section 18.30) for further description.

-LOAD CARDS

SUB-SEA

This card defines the effects of the primary explosion and the subsequent bubble oscillations.

Refer to the LS-DYNA User's Manual (Section 18.26) for further description.

-LOAD CARDS

THERMAL

This card defines nodal temperatures that thermally load the structure.

Refer to the LS-DYNA User's Manual (Section 18.33) for further description.

BOUNDARY CONDITIONS (NASTRAN)

The functions of the BOUNDARY CONDITIONS MENU create and verify constraints and loads on finite element models. The functions in the BOUNDARY CONDITIONS MENU are organized as follows:

BOUNDARY CONDITIONS

CREATE SUBCASE
DELETE SUBCASE
SHOW SUBCASE
MODIFY SUBCASE
LOADING
SPC
INITIAL VELOCITY
EXIT

A detailed description of each function is given in the following sections.

note: If the selected analysis code is LS-DYNA a separate menu with different options will appear. See Section 6.8 BOUNDARY CONDITIONS (LS-DYNA).

CREATE SUBCASE

This function incorporates load sets and SPC sets into a subcase. The user may use any combination of load sets and SPC sets in each subcase.

1. VPG prompts the user to enter a series of identifiers:
   > ENTER SUBCASE NUMBER (XXXX) OR -1 TO EXIT
   - The subcase number defaults to the next available set number in sequence.

   > ENTER TITLE FOR THIS SUBCASE
   - The user enters the name of the subcase.

   > DEFINE SPC SET FOR SUBCASE XXXX

   > SELECT SPC SET
   - The user can choose from a list of all available SPC sets. If there are no SPC sets defined in the database, then VPG will enter the CREATE SPC SET menu and prompt the user to define the SPC set.

2. VPG prompts:
   > SELECT LOAD SET
   - See CREATE LOAD SET later in this section.
   - See DEFINE FORCE later in this section.
   - See DEFINE PRESSURE later in this section.

DELETE SUBCASE

This function deletes selected subcases from the VPG database.

1. VPG prompts:
   > SELECT SUBCASE
   - All available subcases are listed in the options area.

2. Before deleting the subcase, VPG checks by prompting:
   > SPC SET xxxx AND LOAD SET xxxx IN SUBCASE xxxx
   > OK TO DELETE SUBCASE xxxx (Y/N)

3. The subcases are deleted (or not deleted) depending on the user's response.

note: DELETE SUBCASE does not delete the SPC and Load sets from the subcase.

SHOW SUBCASE

This function displays the SPC and Load sets in the selected subcases in the VPG database.

1. VPG prompts:
   > SELECT SUBCASE
   - All available subcases are listed in the options area.

MODIFY SUBCASE

This function modifies selected subcases in the VPG database.

1. VPG prompts:
   > SELECT SUBCASE
   - All available subcases are listed in the options area.

2. Once the user selects a subcase to modify, VPG prompts to select/create a new SPC/Load set number. Directions are given in CREATE SUBCASE and are covered in the previous pages.

LOADING

This menu lists the options to configure and modify load sets in the VPG database.

1. VPG displays the following list of commands in the options area:

```
NEW SET
CURRENT SET
CLEAR FORCE
CLEAR PRESSURE
DELETE SET
DEFINE FORCE
DEFINE PRESSURE
LIST FORCE
LIST PRESSURE
REMOVE FORCE
```

Continued on next page.

```
REMOVE PRESSURE
SHOW FORCE
SHOW PRESSURE
SHOW SPC
VIEW
EXIT
```

2. Each of these commands are described as subsections in the following pages.

-LOADING OPTIONS
NEW SET

This function defines a new load set. The load set may be used in the subcase definition.

1. **NEW SET**
   > ENTER LOAD SET NUMBER (=XXXX) OR -1 TO EXIT
   - The user may enter an integer as the load set number. If a load set with the entered number exists, the user is prompted to enter a different integer.

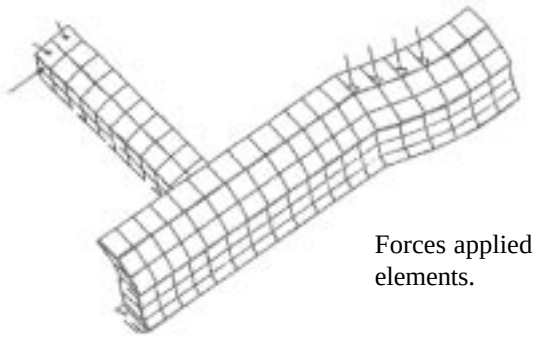
2. VPG prompts:
   > ENTER NODAL FORCE (FX, FY, FZ, MX, MY, MZ) OR E TO EXIT
   - Real numbers or integers are acceptable.

   > DEFINE FORCE LOCATION
   SELECT NODES
   - Nodes may be selected one at a time with the left mouse button.
   - EXIT or DONE will exit the user to the ENTER NODAL FORCE FX, FY, FZ, MX, MY, MZ prompt (step 2). To exit nodal force definitions, enter E for EXIT.

3. VPG prompts:
   > ENTER ELEMENT PRESSURE LOAD OR E TO EXIT
   - This command allows the user to apply pressure loads to the element centers in the VPG model. Distributed element pressure loads are applied normal to the element surface. Positive pressure loads are in the same direction as the positive element normal according to the right hand rule. Negative pressure loads are in the opposite direction of the positive element normal.

Continued on next page.

4. VPG prompts:
   > SELECT ELEMENT FOR PRESSURE LOAD
   > SELECT ELEMENTS
   - The left mouse button may be used to select one element at a time.
   - DONE returns to step 3.
   - EXIT exits the command.
   - ABORT exits the user to the LOADING OPTIONS submenu and disregards previous selections.



Forces applied to nodes and pressures applied to elements.

- LOADING OPTIONS

CURRENT SET

This function makes the desired load set current. The load set may consist of nodal force/moment and pressure loads.

1. VPG prompts:
   > SELECT CURRENT LOAD SET
   - The user can choose from a list of all available load sets.

-LOADING OPTIONS
CLEAR FORCE

This function deletes all NODAL FORCES in the current load set.

1. VPG prompts:
   > O.K. TO REMOVE ALL FORCES? (Y/N)
   ? YES
   NO
   - YES will delete the FORCES on all grids in the current load set. If there is no data in the load set, VPG will prompt:

   > NO FORCE NOR MOMENT EXISTS IN THIS LOAD SET
   O.K. TO DELETE THIS LOAD SET? (Y/N)

-LOADING OPTIONS
CLEAR PRESSURE

This function deletes all PRESSURE LOADS in the current load set.

1. VPG prompts:
   > O.K. TO REMOVE ALL PRESSURE LOADS? (Y/N)
   ? YES
   NO
   - YES will delete the PRESSURE LOADS on all elements in the current load set.

2. If there is no data in the load set, VPG will prompt:
   > NO FORCE NOR MOMENT EXISTS IN THIS LOAD SET
   O.K. TO DELETE THIS LOAD SET? (Y/N)

-LOADING OPTIONS
DELETE SET

This function deletes the selected load set.

1. VPG prompts:
   > SELECT LOAD SET
   - Available load sets are displayed in the options area.

Continued on next page.

2. VPG prompts:
   > O.K. TO DELETE LOAD SET? (Y/N)
   - YES deletes load set.
   - NO aborts operation.
   - If the load set does not contain any loads, VPG prompts:
   > NO LOAD SPECIFIED IN LOAD SET xxxx

-LOADING OPTIONS
DEFINE FORCE

This function defines forces and moments on the displayed model in the current load set.

1. VPG prompts:
   > ENTER NODAL FORCE (FX, FY, FZ, MX, MY, MZ) OR E TO EXIT
   - The details of this procedure are covered in CREATE LOAD SET.

-LOADING OPTIONS
DEFINE PRESSURE

This function defines pressure on selected elements in the current load set.

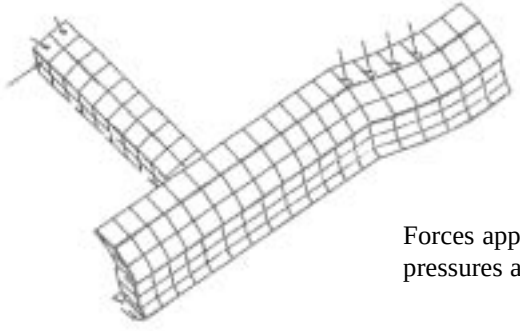
1. VPG prompts:
   > ENTER ELEMENT PRESSURE LOAD OR E TO EXIT
   - The details of this procedure are covered in CREATE LOAD SET.

-LOADING OPTIONS
LIST FORCE

This function displays the applied forces and lists node numbers and respective loads in the command area of the screen.

Continued on next page.

1. VPG prompts:
   > SELECT NODE FOR FORCE
   ? ALL NODES
   - All nodes with applied forces will have arrowheads displayed, but no specific load magnitudes will be listed.
     DONE
     EXIT
     NODE
   - Pressing the left mouse button or typing N will list the applied force of the grid nearest to the cursor.
   - If no FORCES or MOMENTS are applied at the selected grid, VPG will display the message:
     > NO LOAD SPECIFIED AT SELECTED NODE
     and remain in the LIST FORCE option.
   - EXIT or DONE will exit to the LOADING OPTIONS menu.



Forces applied to nodes and pressures applied to elements.

-LOADING OPTIONS
LIST PRESSURE

This function displays the pressure loads and respective element numbers in the command area of the screen.

1. VPG prompts:
   > SELECT ELEMENT FOR PRESSURE LOAD
   ? CURSOR LOCATION

Continued on next page.

DONE
EXIT

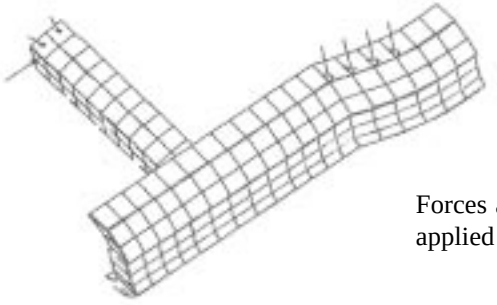
- If no pressure is applied at the selected element, VPG will display the message:

> NO PRESSURE DEFINED AT SELECTED ELEMENT

and remain in the LIST option.

- EXIT or DONE returns to the LOADING OPTIONS menu.
- If the selected element has a pressure load, VPG prompts:

> PRESSURE LOAD AT EL. xxxx = xxxx.xx



Forces applied to nodes and pressures applied to elements.

-LOADING OPTIONS
REMOVE FORCE

This function removes nodal forces from selected grids in the current load set.

1. VPG prompts:

> REMOVE NODAL FORCE
SELECT NODES

- The left mouse button may be used to remove selected nodal forces one at a time.
- When a nodal force is removed, the node is highlighted by a small circle.
- If a node is selected twice, the following message appears:

> NODE xxxx ALREADY PICKED

- EXIT or DONE returns the user to the REMOVE FORCE, SELECT NODES prompt (step 2).
- ABORT returns the user to the REMOVE FORCE submenu without deleting the selected nodal forces.

-LOADING OPTIONS
REMOVE PRESSURE

This function removes pressure loads from selected elements in the current load set.

1. VPG prompts:
   > REMOVE PRESSURE LOAD
   > SELECT ELEMENTS
   - When a pressure load is removed, the element is highlighted and the arrow is removed.
   - If an element is selected more than once, the following message appears:

   > EL xxxx ALREADY PICKED
   > SELECT ELEMENTS
   - EXIT or DONE will exit the user to the LOADING OPTIONS submenu.
   - ABORT will exit the user to the LOADING OPTIONS submenu without removing any of the selected pressure loads.

-LOADING OPTIONS
SHOW FORCE (toggle)

The SHOW FORCE function is a toggle switch that displays the forces (single arrowhead) and moments (double arrowhead) in the current load set.

> FORCE WILL BE DISPLAYED (default)
or
> FORCE WILL NOT BE DISPLAYED

-LOADING OPTIONS
SHOW PRESSURE (toggle)

The SHOW PRESSURE function is a toggle switch that displays the pressure points in the current load set.

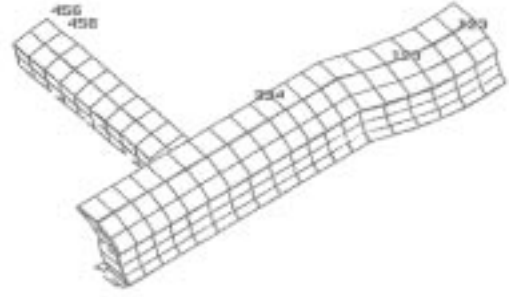
> PRESSURE WILL BE DISPLAYED (default)
or
> PRESSURE WILL NOT BE DISPLAYED

-LOADING OPTIONS
SHOW SPC (toggle)

The SHOW SPC function is a toggle switch that displays the SPC's on the screen in the current load set. (The default status for LOADING OPTIONS is OFF.)

```
>   SPC WILL BE DISPLAYED
```
or
```
>   SPC WILL NOT BE DISPLAYED
```

Constraints on individual nodes are displayed.



SPC

This function allows the user to create and/or modify the SPC (single point constraint) options that constrain nodes in six degrees of freedom.

1. VPG displays a list of additional commands in the options area:

```
NEW SET
CURRENT SET
CLEAR SPC
DELETE SPC
DEFINE DOF
REMOVE SPC
LIST SPC
SHOW SPC
SHOW FORCE
VIEW
EXIT
```

2. Each of these commands is described in the following pages.

- SPC OPTIONS

NEW SET

This function creates a new SPC set. The SPC set may be used later for SPC set definition.

1. FEMB prompts:

   > ENTER SPC SET NUMBER (=XXXX) OR -1 TO EXIT

   - The user must enter an integer before proceeding.

The options in the top and botom of the SPC DEGREE OF FREEDOM box represent alternate methods for defining the degree of freedom. Only the top or the bottom can be selected; not both.

X-Y SYMMETRY

- Assigns symmetric boundary conditions to the XY plane.

Y-Z SYMMETRY

- Assigns symmetric boundary conditions to the YZ plane.

X-Z SYMMETRY

- Assigns symmetric boundary conditions to the XZ plane.

X-Y ANTI-SYMMETRY

- Assigns asymmetric boundary conditions to the XY plane.

Y-Z ANTI-SYMMETRY

- Assigns asymmetric boundary conditions to the YZ plane.

X-Z ANTI-SYMMETRY

- Assigns asymmetric boundary conditions to the XZ plane.

Continued on next page.

X - TRANSLATION
- X-translation will be constrained.

Y - TRANSLATION
- Y-translation will be constrained.

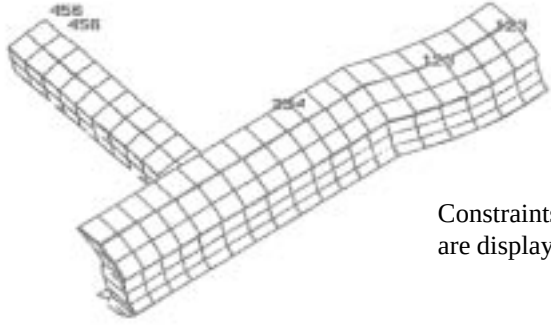
Z - TRANSLATION
- Z-translation will be constrained.

X - ROTATION
- X-rotation will be constrained.

Y - ROTATION
- Y-rotation will be constrained.

Z - ROTATION
- Z-rotation will be constrained.

2. VPG prompts the user to select constrained nodes:
   > D.O.F. xxxx IS FIXED
   > SELECT NODES
   - Nodes may be selected one at a time with the left mouse button.
   - EXIT or DONE will exit the user to the ENTER SPC SET NUMBER prompt (step 1).
   - ABORT will exit the user to the SPC OPTIONS menu without constraining any of the selected grids.



Constraints on individual nodes are displayed.

- SPC OPTIONS
CURRENT SET

This command allows the user to make the selected SPC set active.

1. VPG prompts:
   > SELECT CURRENT SPC SET
   - The user can choose from a list of all available SPC sets.

- SPC OPTIONS
CLEAR SPC

This function deletes all of the constraints in the current SPC set.

1. VPG prompts:
   > O.K. TO CLEAR ALL SPC IN THIS SPC SET? (Y/N)
   - YES clears all SPC's in the set.
   - NO aborts operation.

2. If **YES** is selected, VPG prompts:
   > ALL SPC HAVE BEEN REMOVED

- SPC OPTIONS
DELETE SET

This function deletes the selected SPC set.

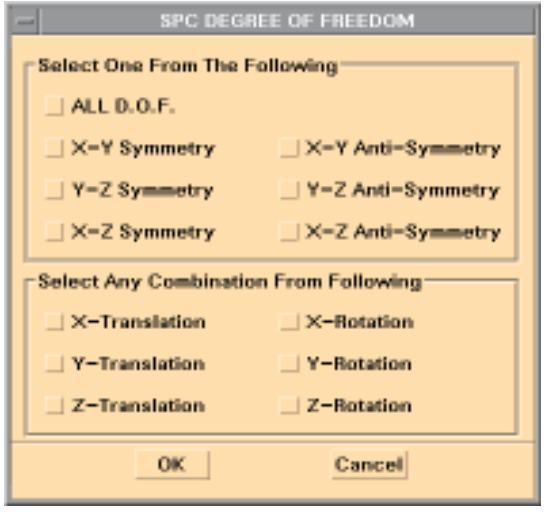
1. VPG prompts:
   > SELECT SPC SET
   - All available SPC sets are displayed for selection in the options area.

2. After a SPC set has been selected for deletion, VPG prompts:
   > O.K. TO DELETE SPC SET xxxx (Y/N)
   - YES deletes the SPC set and VPG prompts:
     > SPC SET xxxx HAS BEEN DELETED
   - NO aborts the command.

- SPC OPTIONS
DEFINE DOF

This function defines SPC's in the current SPC set. After selecting DEFINE , VPG displays the following dialogue box. Once the DOF is determined in the dialogue box, VPG prompts the user to select the nodes. DONE finishes the selection and displays the nodes with their SPC set number.

Continued on next page.



The options in the top and botom of the SPC DEGREE OF FREEDOM box represent alternate methods for defining the degree of freedom. Only the top or the bottom can be selected; not both.

X-Y SYMMETRY
- Assigns symmetric boundary conditions to the XY plane.

Y-Z SYMMETRY
- Assigns symmetric boundary conditions to the YZ plane.

X-Z SYMMETRY
- Assigns symmetric boundary conditions to the XZ plane.

X-Y ANTI-SYMMETRY
- Assigns asymmetric boundary conditions to the XY plane.

Y-Z ANTI-SYMMETRY
- Assigns asymmetric boundary conditions to the YZ plane.

X-Z ANTI-SYMMETRY
- Assigns asymmetric boundary conditions to the XZ plane.

X - TRANSLATION
- X-translation will be constrained.

Y - TRANSLATION
- Y-translation will be constrained.

Z - TRANSLATION
- Z-translation will be constrained.

X - ROTATION
- X-rotation will be constrained.

Y - ROTATION
- Y-rotation will be constrained.

Z - ROTATION
- Z-rotation will be constrained.

- SPC OPTIONS
REMOVE SPC

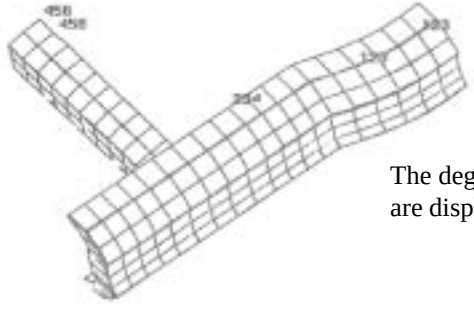
This function removes constraints (deletes SPC's) from selected grids/nodes in the current or selected SPC set.

1. VPG prompts:
   > RELEASE SPC
   SELECT NODES
   - EXIT or DONE will return the user to the RELEASE SPC (step 2).
   - ABORT will return the user to the SPC OPTIONS menu without releasing any of the selected grids.

- SPC OPTIONS
LIST SPC

This function displays the DOF (degrees of freedom) of the selected grids on the screen.

1. VPG prompts:
   > SELECT NODE FOR SPC LIST
   ? ALL NODES
   - All nodes with SPC's will be displayed with the appropriate DOF.
     DONE
     EXIT
     NODE
   - If no DOF are constrained on the selected grid, VPG will display the message:
   > NO SPC SPECIFIED AT SELECTED NODE
   and remain in the LIST SPC option.
   - EXIT or DONE will exit this option.



The degrees of freedom for the nodes are displayed.

- SPC OPTIONS
SHOW SPC (toggle)

This function is a toggle switch that displays the SPC's in the current SPC set.

> SPC WILL BE DISPLAYED (default)
or
> SPC WILL NOT BE DISPLAYED

- SPC OPTIONS
SHOW FORCE (toggle)

The SHOW FORCE function is a toggle switch that displays the forces (single arrowhead) and moments (double arrowhead) in the current load set. The default for the SPC options may be turned off and restored with this command.

> FORCE/PRESSURE WILL BE DISPLAYED
or
> FORCE/PRESSURE WILL NOT BE DISPLAYED

INITIAL VELOCITY

This function allows the user to define and assign both linear and angular velocities to selected nodes.

1. VPG displays a list of additional commands in the options area:

   CLEAR INITIAL VEL.
   DEFINE INITIAL VEL.
   LIST INITIAL VEL.
   REMOVE INITIAL VEL.
   SHOW INITIAL VEL.
   SHOW SPC
   VIEW
   EXIT

2. Each of these commands is described in the following pages.

- INITIAL VELOCITY
CLEAR INITIAL VELOCITY

This function enables the user to remove all initial velocities from the database.

1. VPG prompts:
   > O.K. TO REMOVE ALL INITIAL VELOCITY? (Y/N)
   - YES clears all initial velocities in the set.
   - NO aborts operation.

- INITIAL VELOCITY
DEFINE INITIAL VELOCITY

This function enables the user to define and assign initial velocity to the displayed model.

1. VPG prompts:
   > ENTER INITIAL VELOCITY (VX, VY, VZ, RX, RY, RZ) OR E TO EXIT
   > DEFINE LOCATION FOR INITIAL VELOCITY
   > SELECT NODES

2. Selecting DONE or EXIT executes this command.

- INITIAL VELOCITY
LIST INITIAL VELOCITY

The selected initial velocities of the nodes are listed in the prompt area .

1. VPG prompts:
   > SELECT NODE FOR INITIAL VELOCITY
   ? ALL NODES
   - All nodes with an initial velocity are highlighted with an arrow.
     DONE
     EXIT
     NODE
   - EXIT or DONE terminates this option.

- INITIAL VELOCITY
REMOVE INITIAL VELOCITY

This function enables the user to remove the initial velocity from selected nodes.

1. VPG prompts:
   > SELECT NODES

2. DONE or EXIT executes the command.

- INITIAL VELOCITY
SHOW INITIAL VELOCITY (toggle)

This function is a toggle switch that displays the initial velocities on the screen.

> INITIAL VELOCITY WILL BE DISPLAYED (default)
or
> INITIAL VELOCITY WILL NOT BE DISPLAYED

- INITIAL VELOCITY
SHOW SPC (toggle)

The SHOW SPC function is a toggle switch that displays SPC's on the screen in the current SPC set.

> SPC WILL BE DISPLAYED (default)
or
> SPC WILL NOT BE DISPLAYED

MATERIAL PROPERTY (LS-DYNA)

The functions in the MATERIAL PROPERTY MENU are designed to define and modify the material properties in the database. Parts are displayed according to the color of their material definition. Parts without material definition are displayed in white.

The functions in this menu are organized as follows:

MATERIAL PROPERTY

ASSIGN MATERIAL
CHANGE COLOR
COPY
CREATE
DELETE
DEFINE PROPERTIES
ERASE UNREF. MAT.
LIST
MATERIAL COLOR ON/OFF
ON/OFF
READ MATERIALS
WRITE MATERIALS
THERMAL MATL MENU
VIEW
EXIT

By switching solver code settings in the ANALYSIS PROGRAM command in the SETUP menu, the user can assign and create materials that are solver-code specific. The user may assign and create materials at the beginning or at the end of a VPG session.

A detailed description of each function is given in the following sections.

note: If the analysis code is set to NASTRAN, see section 6.11.

ASSIGN MATERIAL

This function assigns materials to selected parts.

1. If no materials are present in the database, VPG gives the message, "NO MATERIALS DEFINED IN DATABASE" and returns to the command prompt.

2. Otherwise, VPG prompts the user to select a material:

```
>   SELECT MATERIAL
>   PICK AN ELEM(C) OR PROPERTY NAME(P) OF A MATERIAL
?   CURSOR AT ELEMENT
    EXIT
    PROPERTY NAME
    UNDO SELECTION
```

   - Users may select an existing material by entering one by name or by picking an element on the screen that is assigned to the desired material (default).

3. After the user selects the materials, VPG prompts the user to assign parts to the material.

4. The material will be assigned to the selected elements of the parts. The selected parts will be shown in the material color after the user exits this command.

CHANGE COLOR

This function changes the color of a selected material.

1. The color bar is displayed on the right edge of the graphic area.

2. VPG prompts the user to select a material:

```
>   SELECT MATERIAL TO CHANGE COLOR
>   PICK AN ELEM(C) OR PROPERTY NAME(P) OF A MATERIAL
?   CURSOR AT ELEMENT
    MATERIAL NAME
```

Continued on next page.

UNDO SELECTION
EXIT

3. VPG prompts the user to select a color from the color bar.

4. The color of the material changes to the user-selected color from the color bar.

COPY

This function duplicates the properties from one material onto another material.

1. VPG displays the material card list and prompts:
   > SELECT CARD TO COPY

2. Once the user selects a card to copy VPG copies the material and prompts:
   > MATERIAL ID [x] DUPLICATED FROM MATERIAL ID [x]

CREATE

This function creates a new material.

1. After selecting create, VPG offers the folllowing material categories:
   STRUCTURAL
   DAMPER/SPRING

2. Selecting either category yields an LS-DYNA material card list. When the desired material is selected from the list, VPG displays its definition card(s). The user can then edit the material card for desired properties.

DELETE MATERIALS

This function deletes the user-selected materials from the database. VPG displays the material card list. Selected materials are marked with an asterisk. When the user exits the material card list, the selected material are deleted.

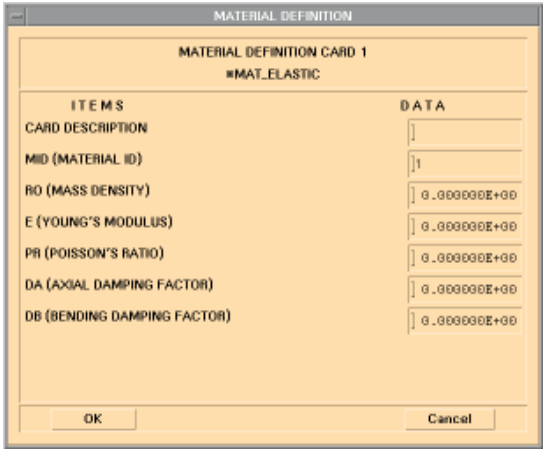
DEFINE PROPERTIES

This function defines or modifies the properties of a selected material.

1. VPG prompts the user to select a material.
   > PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL

2. After the user selects a material, its MATERIAL DEFINITION CARD appears.

3. The user may modify the properties and/or change the values manually in the MATERIAL DEFINITION CARD.



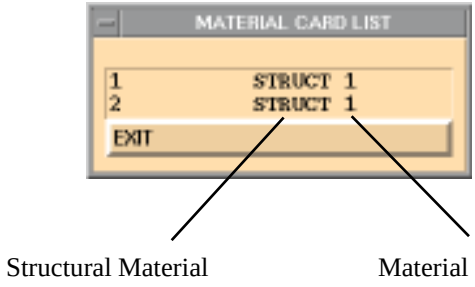
Define Properties

ERASE UNREFERENCED MATERIAL

This function automatically deletes all materials that have no parts assigned to them.

LIST MATERIALS

This function lists the defined materials in the MATERIAL CARD LIST.



MATERIAL COLOR ON/OFF

This function toggles the display of parts between the material and part color.

ON/OFF

This function toggles the selected material on or off.

1. VPG prompts the user to select a material.
   > PICK ELEM (C), OR PROPERTY NAME (P) OF A MATERIAL

2. The selected material and parts are turned on/off.

RENUMBER

This function allows the user to assign a a new material ID (MID) number to a material type .

THERMAL MATERIAL MENU

This function creates thermal materials.

1. VPG prompts:

```
>  SELECT OPTION
   ASSIGN MATERIAL
   CREATE
   DELETE
   DEFINE PROPERTIES
   LIST
   VIEW
```

   - The functions of the THERMAL MATERIAL MENU are similar to the functions of the MATERIAL PROPERTY MENU. See the preceding pages for a complete description of these functions.

MATERIAL PROPERTY (NASTRAN)

The functions in the MATERIAL PROPERTY MENU are designed to define and modify the material properties in the database. Parts are displayed according to the color of their material definition. Parts without material definition are displayed in white.

The functions in this menu are organized as follows:

MATERIAL PROPERTY

ASSIGN MATERIAL
CHANGE COLOR
COPY
CREATE
DELETE
DEFINE PROPERTIES
LIST
MATERIAL PART COLOR
ON/OFF
RENAME
READ MATERIALS
RENUMBER ID
SHOW STRAIN-STRESS
WRITE MATERIALS
VIEW
EXIT

By switching solver code settings in the ANALYSIS PROGRAM command in the SETUP menu, the user can assign and create materials that are solver-code specific. The user may assign and create materials at the beginning or at the end of a VPG session.

A detailed description of each function is given in the following sections.

ASSIGN MATERIAL

This function assigns materials to selected parts.

1. If no materials are present in the database, VPG gives the message, "NO MATERIALS DEFINED IN DATABASE" and returns to the command prompt.

2. Otherwise, VPG prompts the user to select a material:
   ```
   >   SELECT MATERIAL
   >   PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL
   ?   CURSOR AT ELEMENT
       MATERIAL NAME
       UNDO SELECTION
       EXIT
   ```
   - Users may select an existing material by entering one by name or by picking an element on the screen that is assigned to the desired material (default).

3. After the user selects the materials, VPG prompts the user to assign parts to the material.

4. The material will be assigned to the selected elements of the parts. The selected parts will be shown in the material color after the user exits this command.

CHANGE COLOR

This function changes the color of a selected material.

1. The color bar is displayed on the right edge of the graphic area.

2. VPG prompts the user to select a material:
   ```
   >   SELECT MATERIAL TO CHANGE COLOR
   >   PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL
   ?   CURSOR AT ELEMENT
       MATERIAL NAME
       UNDO SELECTION
       EXIT
   ```

Continued on next page.

3. VPG prompts the user to select a color from the color bar.

4. The color of the material changes to the user-selected color from the color bar.

COPY

This function duplicates the properties from one material onto another material.

1. VPG prompts the user for the name of the target material.
   > SELECT NAME OF MATERIAL TO BE DUPLICATED
   > PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL

2. Once the user selects the material, VPG prompts:
   > MATERIAL [displayed material name] IS SELECTED
   > ENTER NAME OF DUPLICATE MATERIAL
   - The user enters the name of the duplicate material.

3. If the name already exists, VPG prompts:
   > XXXX ALREADY EXISTS, OK TO COPY PROPERTIES TO? (Y/N)
   - YES - Material is copied as indicated.
   - NO - Returns to the MATERIAL MENU.

CREATE

This function creates a new material.

1. VPG prompts for the new material name.
   > ENTER NAME OF NEW MATERIAL OR "EX" TO EXIT
   - VPG runs a check for the name in the database. If the name already exists, VPG prompts for a different name.

2. After entering the material name, a pop-up window appears with the material types. The list of the material types depends on the analysis program (e.g., NASTRAN, LS-DYNA) defined in the SETUP menu.

3. Once the user has chosen a material type, a Pop Up window appears with the material's properties. VPG prompts:
   > DEFINE/MODIFY DATA
   - The displayed property table allows the user to insert values for the material's properties.

DELETE MATERIALS

This function deletes the user-selected materials from the database.

1. VPG prompts the user for the name of the target material.
   ```
   >   DE-DELETE MATERIALS
   >   PICK AN ELEM (C), OR MATERIAL NAME(M) OF A MAT
   ```

2. The user selects the material to be deleted. The elements that are assigned with that material turn gray, and VPG prompts:
   ```
   >   MATERIAL [name] ASSIGNED TO  XXXX PARTS
   >   OK TO  DELETE MATERIAL [name] ?
   ?   YES
       NO
   ```
   - If the user enters YES, VPG deletes the material and changes the color of the part from gray to white. The white indicates that the part does not have any material properties assigned to it.
   - If the user enters NO, VPG retains the material and displays the parts in the color of the original material.

DEFINE PROPERTIES

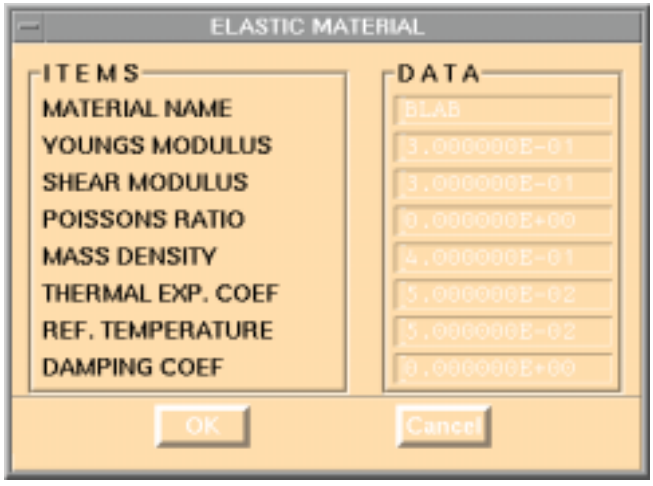
This function defines or modifies the properties of a selected material.

1. VPG prompts the user to select a material.
   ```
   >   PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL
   ```

2. After the user selects a material, the MATERIAL DEFINITION window appears and a prompt reads:
   ```
   >   DEFINE/MODIFY DATA
   ```

3. VPG displays the properties and the type of material that the user has defined in a table. The user may modify the properties and/or change the values manually in the window.

Continued on next page.



Define Properties

LIST MATERIALS

This function lists the defined materials in the MATERIAL LIST.

MATERIAL PART COLOR

This function toggles the display of parts between the material and part color.

ON/OFF

This function toggles the selected material on or off.

1. VPG prompts the user to select a material.
   > PICK ELEM (C), OR MATERIAL NAME (M) OF A MATERIAL

2. The selected material and parts are turned on/off.

RENAME

This function renames the selected material.

1. VPG prompts the user to select a material.
   > SELECT MATERIAL TO RENAME
   > PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL

2. The user selects a material to be renamed, then VPG prompts:
   > MATERIAL [name] IS SELECTED
   > ENTER NEW NAME OR "EX" TO EXIT

3. VPG prompts the user to enter a new material name. If the name exists in the database, VPG prompts for a different name.

READ MATERIALS

This function reads materials from the material library.

RENUMBER ID

This function renumbers the material ID for the selected material or all materials.

1. VPG prompts for the option:
   ```
   >   RENUMBER MID
   >   SELECTED MATERIAL OR ALL? (S/A)
   ?   SELECT MATERIAL
       ALL MATERIALS
       EXIT
   ```

2. If the user enters **SELECT MATERIAL**, VPG prompts:
   ```
   >   PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL
   ```

3. After the user enters the material, VPG prompts for a new ID.
   ```
   >   MATERIAL XXXX IS SELECTED.
   >   ENTER MID [number] FOR MAT [name]
   ```

4. If the user selects **ALL MATERIALS**, VPG prompts for a new ID for each material. The following prompt will be repeated for each material in the database.
   ```
   >   ENTER MID [number] FOR MAT [name] OR E TO EXIT
   ```
   - If the user enters nothing, VPG defaults to the original values.

SHOW STRAIN-STRESS

This function plots the strain-stress curve for the selected materials.

1. VPG prompts the user to select material(s):
   ```
   >   SELECT MATERIAL FOR STRAIN-STRESS CURVE
   >   PICK AN ELEM(C) OR MATERIAL NAME(M) OF A MATERIAL
   ```
   - Exit takes the user to step four once a material is selected.

2. The user selects materials for plotting. If the user selects a rigid material, VPG issues the warning message:
   ```
       NO STRAIN-STRESS CURVE FOR RIGID MATERIAL
   ```

Continued on next page.

3. The strain-stress curves are plotted in the graphic area for the selected materials. The editing functions are listed in the command area. Upon selection of the desired materials, the user chooses one of the following options.

   ? DEFINE LOCATION
   - The user may define a stress-strain curve location by drag window.

   FULL SCREEN
   - The stress-strain curve occupies the entire window.

   EXIT

4. VPG prompts:

   ? GRAPH LOCATION
   CURVE OPERATIONS
   CLEAR SCREEN
   DELETE CURVES
   HARDCOPY
   OPTIONS
   QUERY
   RANGE
   SAVE CURVES
   EXIT

   - For a description of these commands, see the TIME HISTORY section of the manual.

5. Once the user exits this function, the original structure display will resume in the graphic area.

WRITE MATERIAL

After creating a material this function assigns it to a user-defined material library.

ELEMENT PROPERTY

The functions in the ELEMENT PROPERTY MENU are designed to define and modify the element properties (physical) in the database. When using these commands, parts with property definition are displayed according to their property colors. Parts without property definition are shown in white.

The functions in the ELEMENT PROPERTY MENU are organized as follows:

ELEMENT PROPERTY

ASSIGN PROPERTY/SECTION
CHANGE COLOR
COPY
CREATE
DELETE
DEFINE PROPERTY/SECTION
LIST
ON/OFF
PROPERTY/SECTION PART COLOR
RENAME
RENUMBER ID
VIEW
EXIT

By toggling the solver code settings in the ANALYSIS PROGRAM command in the SETUP window, the user can assign and create properties that are solver-code specific. However, the element property menu and its functions will change slightly between NASTRAN and LS-DYNA analysis. For example, NASTRAN refers to element properties where LS-DYNA refers to element sections.

The user may assign and create materials at the beginning or at the end of a VPG session.

A detailed description of each function is given in the following sections.

ASSIGN PROPERTY

This function assigns element properties (physical) to selected parts.

1. VPG prompts the user to select a property:
   > PICK AN ELEM(C) OR PROPERTY NAME(P) OF A PROPERTY
   - The user may select a property by picking an element on the screen that contains a desired property (default) or by selecting a property name
   - If no properties are present in the database, the user will be prompted with the message, "NO PROPERTIES DEFINED IN DATABASE" and the command prompt will return.

2. VPG prompts the user to designate a part(s) to accept the selected property.
   > SELECT PART TO ASSIGN PROPERTY
   > PICK AN ELEMENT OR A PART NAME OF A PART

3. The property will be assigned to the elements in the selected parts. After the user exits, the selected parts will be shown in the property color.

CHANGE COLOR

This function changes the color of a selected property.

1. The color bar is displayed at the right edge of the graphic area.

2. VPG prompts the user to select a property.
   > SELECT PROPERTY TO CHANGE COLOR
   > PICK AN ELEM(C) OR PROPERTY NAME(P) OF A PROPERTY

3. After the user selects a property, the prompt reads:
   > PROPERTY [name] IS SELECTED
   SELECT COLOR FROM COLOR BAR OR EXIT

4. The new color is assigned to the selected property.

COPY (NASTRAN)

This function copies the properties from one element to another.

Continued on next page.

1. VPG prompts the user to select a property.
   > SELECT NAME OF PROPERTY TO BE DUPLICATED
   > PICK AN ELEM(C) OR PROPERTY NAME(P) OF A PROPERTY

2. After the user selects the desired property to copy, VPG prompts:
   > PROPERTY [name] IS SELECTED
   > ENTER NAME OF DUPLICATE PROPERTY

3. Once the name is entered, VPG assigns the copied property to it and prompts:
   > PROPERTY [name] DUPLICATED FROM PROPERTY [name]

COPY (LS-DYNA)

This function copies the properties from one element to another.

1. VPG displays the SECTION CARD LIST and prompts the user to select a section:
   > SELECT CARD TO COPY

2. After the user selects the desired section to copy, VPG prompts:
   > SECTION ID [x] DUPLICATED FROM SECTION ID [x]

CREATE (NASTRAN)

This function allows the user to create/define a property.

1. VPG prompts for the new property name.
   > ENTER NAME OF NEW PROPERTY OR "EX" TO EXIT
   - VPG checks the database for the name of the property. If the name already exists, VPG prompts the user to enter a different name.

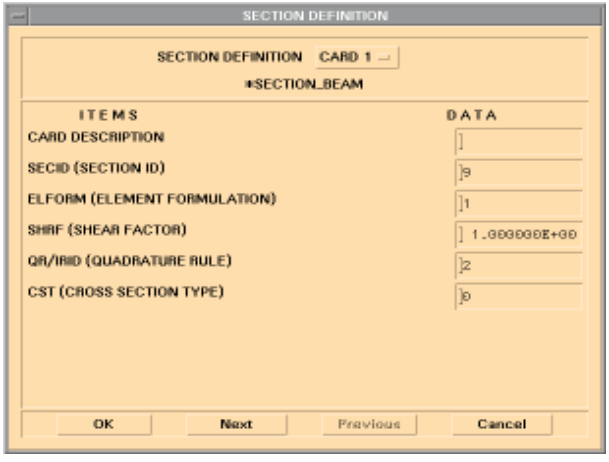
2. After the user enters the appropriate property name, VPG displays a list property types and prompts:
   > SELECT PROPERTY TYPE

Continued on next page.

3. Once a property is selected, VPG displays a definition card in which the user may insert the values for the element properties.

CREATE (LS-DYNA)

This function allows the user to create/define a property. To create a property, VPG first displays a list of section types. Once a type is selected, its definition card is displayed and the user can edit the card for the desired properties.



DELETE (NASTRAN)

This function deletes selected element properties from the database.

1. VPG prompts the user for the target property name.

```
>   DE-DELETE PROPERTY
>   PICK AN ELEM(C), OR PROPERTY NAME(P) OF A PROP.
    [LIST OF PROPERTIES]
?   CURSOR AT ELEMENT
    EXIT
```

2. After the part is selected, VPG displays the part in gray. The following message is issued:

```
>   PROPERTY [name] ASSIGNED TO [number] PARTS
>   OK TO  DELETE PROPERTY [name] ?
?   YES
    NO
```

   - If the user enters YES, VPG deletes the property and changes the color of the part from gray to white. White indicates that the part does not have properties assigned to it.
   - If the user enters NO, VPG retains the original property and displays the part in the color of the original property.

DELETE (LS-DYNA)

This function deletes selected element sections from the database. Sections are displayed in the section card list. Selecting a section places and asterisk by its name. When the user exits the menu, all asterisked sections will be deleted. The following menu options allow the user to select the desired sections:

SECTION NAME (select section by name; also displays section information in dialogue window)
CURSOR AT ELEMENT (select section by element - default)
WINDOW (select part by drag window)
MULTI PT REGION (select part by user defined multi point region)
SELECT ALL
UNDO ALL
UNDO (rejects last)

DEFINE PROPERTIES (NASTRAN)

This function allows the user to define or modify the selected property.

1. VPG prompts the user to select a property.
   > PICK AN ELEM(C) OR PROPERTY NAME(P) OF A PROPERTY

2. After the user selects a property, VPG displays a definition card for editing the element property and prompts:
   > PROPERTY [name] IS SELECTED

3. VPG displays the user-defined properties in a table. The user may manually modify the properties and/or their values in the definition card.

DEFINE SECTION (LS-DYNA)

This function allows the user to define or modify the selected property.

1. VPG displays the section card list with all defined properties.

2. Once the user selects a section to be redefined, VPG displays its definition card. The user can then redefined that section.

ERASE UNREFERENCED SECTION

This function automatically deletes all sections that have no parts assigned to them.

LIST (NASTRAN)

This function lists the names of the element properties in the VPG MENU WINDOW. VPG displays the element properties in their corresponding colors when the properties are turned ON and in white when the properties are turned OFF.

LIST (LS-DYNA)

This function displays the VPG section card list.

ON/OFF

This function turns off the selected property and its parts.

1. VPG prompts the user to select a property.
   ```
   >   PICK AN ELEM (C) OR PROPERTY NAME(P) OF A PROPERTY
       CURSOR AT ELEMENT
       ALL PROPERTIES
       UNDO
       EXIT
   ```

2. VPG turns off the selected property and its parts.

PROPERTY/PART COLOR (toggle)

This function toggles the display of the part between its property and part color.

RENAME

This function renames the selected property.

1. VPG prompts the user to select a property:
   ```
   >   SELECT PROPERTY TO RENAME
   >   PICK AN ELEM(C) OR PROPERTY NAME(P) OF A PROPERTY
   ```

2. After the user selects the property to be renamed, VPG prompts:
   ```
   >   PROPERTY [name] IS SELECTED
   >   ENTER NEW NAME OR "EX" TO EXIT
   ```

3. VPG prompts the user to enter a name for the new property. If the name already exists in the database, then VPG will prompt the user to choose a different name.

RENUMBER PRID (Property ID)

This function renumbers the ID for a selected property or for all properties.

1. VPG prompts:
   > RENUMBER PRID
   > SELECTED PROPERTY OR ALL? (S/A)
   ? SELECT PROPERTY
   ALL PROPERTIES
   EXIT

2. If the user enters SELECT PROPERTY, VPG prompts:
   > ENTER PART NAME OR EXIT
   > PICK AN ELEM(C) OR PROPERTY NAME(P) OF A PROPERTY

3. After the user selects a property, VPG prompts for a new property ID.
   > **ENTER PRID (= [number]) FOR PROP [name]**

4. If the user selects ALL PROPERTIES, VPG will prompt for a new ID for each property. The following prompt will be repeated for each property in the database:
   > ENTER PRID [number] FOR PROP. [name]
   - If the user does not enter a number, VPG retains the original values as a default.

CONTACT INTERFACE

The functions in the CONTACT INTERFACE MENU define and modify contact interface data (sliding and stonewall) for transient nonlinear analysis codes. The functions are organized as follows:

CONTACT DEFINITION

COPY
CREATE
DELETE
MODIFY
LIST
SHOW
EXIT

Once the CONTACT INTERFACE MENU is selected, the active (ON) interfaces are displayed with the active parts. The master segments are drawn as interior outlines of the elements where the segments are defined. The letter “M” is labeled in the middle of the master segments. Likewise, the slave segments are labeled with the letter “S.” The stonewalls are drawn as rectangular shapes and the slave nodes are labeled with small color- filled squares. The segments, slave nodes, and stonewalls are plotted in the interface color. The segment letters for master and slave may be turned OFF by setting the control button SEGMENT SYMBOL in SETUP dialog box.

A detailed description of each function is given in the following sections.

note: Contact interface definition is not required for a NASTRAN analysis.

COPY

This function allows the user to copy previously defined interfaces.

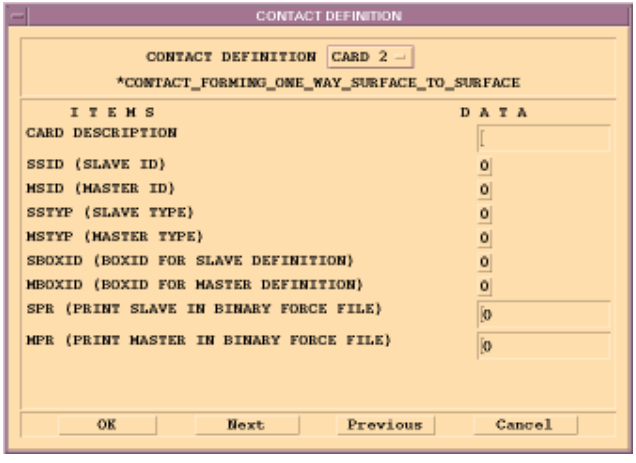
1. VPG displays the CONTACT CARD LIST which lists the defined interfaces and prompts:
   > SELECT CARD TO COPY

2. Upon selecting an interface card from the menu, VPG prompts:
   > CONTACT ID XXXXXXX DUPLICATED FROM CONTACT ID XXXXXXX

CREATE

This function allows the user to define new interfaces.

1. VPG prompts:
   > SELECT CONTACT OPTION
   ? ENTITY
   INTERIOR
   1 DIMENSIONAL
   2 DIMENSIONAL
   3 DIMENSIONAL
   RIGID WALL
   EXIT
   - VPG supports all contact types in LS-DYNA. If general 3D contact type is selected, the program will display all six cards required to define contact. The following notes apply to card 2.



Continued on next page.

- Unnecessary data fields are in gray and unselectable.
- Default Slave and Master segment type is Segment Set (0). The user needs to define the segment type before defining the Slave and Master Set ID.
- If the Segment Type is modified, the corresponding Segment Set ID needs to be redefined.

DELETE

This function deletes previously defined interfaces.

1. VPG prompts:
   > SELECT CARD TO DELETE
   - VPG displays the defined interfaces in the CONTACT CARD LIST with the options:

     > SELECT ALL
     ABORT
     UNDO
     EXIT
   - Cursor pick marks the selected interfaces with an asterisk. Exiting the CONTACT CARD LIST will delete all asterisked interfaces.

MODIFY

This function allows the user to redefine interface properties. Upon selection of the MODIFY command, the CONTACT CARD LIST is displayed with a list of the interfaces. An interface is selected by cursor pick and its definition card is then displayed for the user to modify.

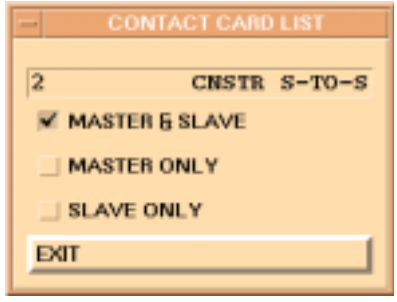
LIST

This function lists the defined interfaces in the CONTACT CARD LIST.

SHOW

This function allows the user to highlight defined interfaces. Upon selection of the SHOW command, the CONTACT CARD LIST is displayed. When the desired interface is selected it will be highlighted in the DRAWING WINDOW. If the desired interface cannot be displayed. VPG prompts:

> THIS CONTACT TYPE CANNOT BE SHOWN



note: The displayed contact may be erased by clicking the CLEAR button in the DISPLAY PARAMETER OPTIONS WINDOW.

DYNA MISCELLANEOUS

The DYNA MISCELLANEOUS MENU contains distinct LS-DYNA specific functions. VPG defines these functions in two ways; graphically when defining such things as a box or vector and with cards based on the definition cards found in the *LS-DYNA Keyword User's Manual*. An example of a definition card can be found in Section 2.13.

AIRBAGS
ALE
CONTROL CARDS
DAMPING CARDS
DATABASE
DEFINE
DEFORMABLE TO RIGID
EQUATION OF STATE
HOURGLASS
INTERFACE CARDS
SEATBELT
TERMINATION

note: The DYNA MISCELLANEOUS option does not appear in the PRE PROCESSING MENU when the analysis code is NASTRAN.

note: All of these options share this standard submenu except CONTROL CARDS. Only the CREATE command differs with respect to each function and will be explained in the following pages.

CREATE

- Defines a new card.

DELETE

- Deletes a selected card or cards from those displayed in a pop-up window.

LIST

- Lists the cards in the option's database via pop-up window.

MODIFY

- Similar to the CREATE command this command lists the defined cards in a pop-up window and allows the user to select and modify them.

EXIT

AIRBAGS

This function provides various cards for defining the thermodynamic behavior of gas flow into an airbag as well as a reference configuration for the fully inflated bag.

1. When CREATE is selected, VPG prompts:

```
>   SELECT OPTION
    ADIABATIC
    HYBRID
    HYBRID JETTING
    LINEAR FLUID
    LOAD CURVE
    SIMPLE AIRBAG
    SIMPLE PRESS. -VOL
    WANG-NEFSKE
    WANG-HEFSKE JET
    W-N MULTI JET
    BAG INTERACTION
    REFERENCE GEOMETRY
    EXIT
```

   - The preceding commands yield varying definition cards. Refer to the *LS-DYNA Keyword User's Manual* (Section 1) for further description.

ALE

This function sets the default control parameters for Arbitrary Lagrange-Eulerian and Eulerian calculations.

1. When **CREATE** is selected, VPG prompts:

```
>   SELECT OPTION
    MULTI-MAT'L PART
    MULTI-MAT'L SET
    SMOOTHING
```

   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 6.8) for further description.

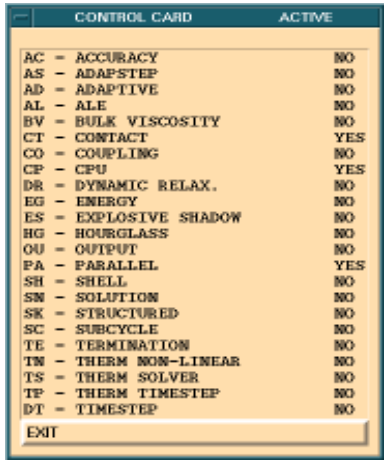
CONTROL CARDS

Control cards are optional default settings.

1. VPG prompts:

```
>  SELECT OPTION
   DEFINE
   LIST
   REMOVE
```

- The command DEFINE reveals the CONTROL CARD LIST and allows the user to define new control cards and modify previously defined control cards
- The command LIST displays the control card list.
- The command REMOVE switches cards from active (YES) to inactive (NO).



A YES in the right column indicates that this type of control card has been defined.

A NO in the right column indicates that this type of control card has not been defined.

DAMPING CARDS

Damping cards define mass weighted nodal damping that applies globally to the nodes of deformable bodies.

Continued on next page.

1. When CREATE is selected, VPG prompts:

   > SELECT OPTION
   GLOBAL
   PART MASS
   PART STIFFNESS

   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 7) for further description.

DATABASE

Database definition cards are used to obtain output result files.

1. VPG prompts:

   > SELECT DATABASE TYPE
   ASCII
   BINARY
   CROSS SECTION
   EXTENT
   HISTORY
   NODAL FORCE GR
   SPRING FORWARD
   SUPER PLASTIC FORMING
   TRACER
   EXIT

note: All these commands have the same following submenu except BINARY:

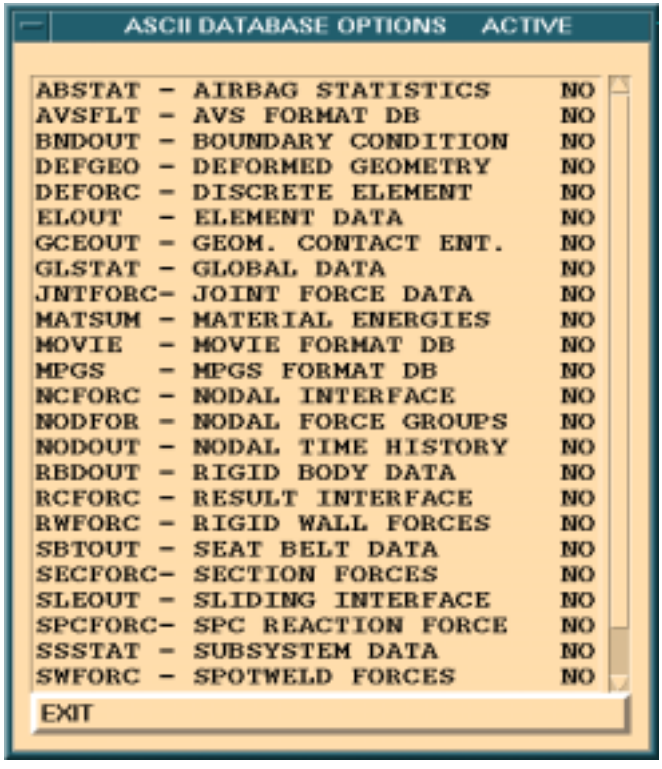
   CREATE
   DELETE
   LIST
   MODIFY
   EXIT

   - Refer to page 1 of this section for a description of these commands.

-DATABASE
ASCII

The definition cards in the ASCII subsection specify the ASCII database output files.

1. When DEFINE is selected, VPG prompts:
   > SELECT OPTION


ASCII DATABASE OPTIONS ACTIVE
ABSTAT - AIRBAG STATISTICS NO
AVSFLT - AVS FORMAT DB NO
BNDOUT - BOUNDARY CONDITION NO
DEFGEO - DEFORMED GEOMETRY NO
DEFORC - DISCRETE ELEMENT NO
ELOUT - ELEMENT DATA NO
GCEOUT - GEOM. CONTACT ENT. NO
GLSTAT - GLOBAL DATA NO
JNTFORC- JOINT FORCE DATA NO
MATSUM - MATERIAL ENERGIES NO
MOVIE - MOVIE FORMAT DB NO
MPGS - MPGS FORMAT DB NO
NCFORC - NODAL INTERFACE NO
NODFOR - NODAL FORCE GROUPS NO
NODOUT - NODAL TIME HISTORY NO
RBDOUT - RIGID BODY DATA NO
RCFORC - RESULT INTERFACE NO
RWFORC - RIGID WALL FORCES NO
SBTOUT - SEAT BELT DATA NO
SECFORC- SECTION FORCES NO
SLEOUT - SLIDING INTERFACE NO
SPCFORC- SPC REACTION FORCE NO
SSSTAT - SUBSYSTEM DATA NO
SWFORC - SPOTWELD FORCES NO
EXIT

2. The user then selects the type of ASCII card to define. VPG displays the definition card to be edited. The time interval, once entered, will be used as the default value for future cards defined. To edit the time interval for subsequently defined cards the user must click once on the card to turn it on and again to have it displayed.

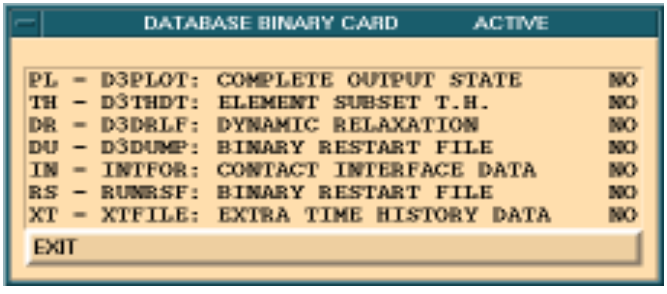
- A YES in the right column of the DATABASE ASCII indicates that this type of card has been defined.
- A NO in the right column indicates that this type of card has not been defined.
- Refer to the LS-DYNA User's Manual (Section 8.4) for further description of the DATABASE ASCII definition cards.

-DATABASE
BINARY

The definition cards in the BINARY subsection specify the BINARY database output files.

1. When DEFINE is selected VPG displays the DATABASE BINARY CARD and prompts:
   > SELECT CARD TO DEFINE

2. The user then selects the type of BINARY card to define. VPG displays the definition card to be edited. The time interval, once entered, will be used as the default value for future cards defined. To edit the time interval for subsequently defined cards the user must click once on the card to turn it on and again to have it displayed.
   - A YES in the right column of the DATABASE BINARY CARD indicates that this type of card has been defined.
   - A NO in the right column indicates that this type of card has not been defined.
   - Refer to the LS-DYNA User's Manual (Section 8.4) for further description of the DATABASE BINARY definition cards.


DATABASE BINARY CARD
ACTIVE
PL - D3PLOT: COMPLETE OUTPUT STATE NO
TH - D3THDT: ELEMENT SUBSET T.H. NO
DR - D3DRLF: DYNAMIC RELAXATION NO
DU - D3DUMP: BINARY RESTART FILE NO
IN - INTFOR: CONTACT INTERFACE DATA NO
RS - RUNRSF: BINARY RESTART FILE NO
XT - XTFILE: EXTRA TIME HISTORY DATA NO
EXIT

-DATABASE
CROSS SECTION

The definition cards in the CROSS SECTION subsection define a cross section for resultant forces written to an ASCII Section Force file.

1. When CREATE is selected, VPG prompts:
   > ENTER NEW CROSS SECTION ID XXXX OR E TO EXIT
   > SELECT CROSS SECTION TYPE
   BY PLANE (see step 2)
   BY SET (see step 6)

2. If the user selects BY PLANE, VPG will first prompt for a part set (if the user wants the cross section only for certain parts of the model). Entering EXIT (without selecting any part set) will create a cross section for the entire model. VPG then prompts for a coordinate system for the cross section (see *Getting Started, Section 3.5*)

3. Next, the user will be prompted for the origin of the cross section.

4 VPG then prompts:
   > ENTER LENGTH L(ALONG U) AND M (ALONG V). 0 FOR INFINITE

5. After entering the U,V increments, VPG displays the cross section. The following commands are available for creating and altering new cross sections:
   CREATE (defines a new cross section)
   DELETE (deletes a defined cross section)
   DISPLAY ON/OFF (toggles the cross section display on/off)
   MODIFY (redefines a cross section)
   SHOW SECTION CUT (lists the defined cross sections in the VPG menu window)
   VIEW
   EXIT

6. If the user selects BY SET, VPG prompts for a set type. If the type of set selected has not yet been defined theuser will be prompted to create one (see SET MENU, Section 6.5, 6.6) Once the set has been determined, follow steps 2-5.

note: Refer to the LS-DYNA User's Manual (Section 8.6) for further description.

-DATABASE
EXTENT

The definition cards in the BINARY subsection specify the BINARY database output files.

1. VPG prompts:
   ```
   >   SELECT OPTION
       CREATE
       DELETE
       LIST
       MODIFY
       EXIT
   ```
   - Refer to page 1 of this section for a description of these commands.

2. When CREATE is selected, VPG prompts:
   ```
   >   SELECT OPTION
       AVS
       BINARY
       MOVIE
       MPGS
       SSSTAT
       EXIT
   ```
3. When the option is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 8.23) for further description.

-DATABASE
HISTORY

The definition cards in the BINARY subsection specify which nodes or elements are output into the binary history file.

1. When CREATE is selected, VPG prompts:
   ```
   >   SELECT OPTION
       BEAM
       BEAM SET
       NODE
       NODE SET
   ```

Continued on next page.

PLATE
PLATE SET
SOLID
SOLID SET
TSHELL
TSHELL SET
EXIT

- The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 8.19) for further description.

-DATABASE
NODAL FORCE GROUP

The definition cards in the BINARY subsection specify a nodal force group for output into ASCII files.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 8.20) for further description.

-DATABASE
SPRING FORWARD

The definition card in the SPRING FORWARD subsection creates a spring forward nodal force file.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 8.21) for further description.

-DATABASE
SUPERPLASTIC FORMING

The definition card in the SUPERPLASTIC FORMING subsection specifies the output intervals to the superplastic forming output files.

1. When DEFINE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 8.22) for further description.

-DATABASE
TRACER

The definition card in the TRACER subsection allows the user to create tracer particles which will save a history of either a material point or a spatial point into an ASCII file; TRHST.

1. When DEFINE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 8.23) for further description.

DEFINE CARDS

The DEFINE definition cards allow the user to define boxes, coordinate systems, load curves, tables, and orientation vectors for various uses.

1. VPG prompts:

```
>   SELECT DATABASE TYPE
    BOX
    COORD. SYSTEM
    CURVE
    SPRING ORIENTATION
    REMOVE TABLE
    VECTOR
    EXIT
```

note: All these commands have the same following submenu except BOX

```
    CREATE
    DELETE
```

Continued on next page.

LIST
MODIFY
EXIT

- Refer to page 1 of this section for a description of these commands.

-DEFINE CARDS
BOX

Defines a specific box shaped volume from two corner points specified in global coordinates.

1. VPG prompts:
   > SELECT OPTION
   CREATE
   - This function creates a new box after the user enters a box ID number and defines the two opposing corners using either nodes, points or xyz coordinates.

   DELETE
   - This function deletes selected boxes. Boxes can be selected via the following commands: CURSOR AT BOX, KEY IN BOX ID, MULTI-POINT REGION, SELECT ALL BOXES, and WINDOW. Boxes that are selected are displayed in gray. When the user exits the menu, they are deleted.

   DISPLAY BOX ON/OFF
   - Toggle switch.

   LIST BOXES
   - Lists the box ID numbers in a pop-up window.

   MODIFY BOX
   - Allows the user to modify a selected box with new point, node, or XYZ coordinates. This command also displays the U, L coordinates repre senting the two corners of the box in global coordinates.

   SHOW CONTENT
   - Displays the nodes and elements of a box in white. The box may be selected by cursor pick or by box ID.

   VIEWING OPTIONS
   - Sends the user to the VIEWING OPTIONS MENU.

   EXIT

2. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 9.2) for further description.

-DEFINE CARDS
COORDINATE SYSTEM

This function allows the user to define a local coordinate system using nodes, points, or vectors.

1. When CREATE is selected, VPG prompts:
   > SELECT OPTION
   NODES
   - Defines a local coordinate system using three nodes.

   SYSTEM
   - Defines a local coordinate system from a previously defined coordinate system or key entry.

   VECTOR
   - Uses a vector to define the coordinate system.

   - When using the MODIFY function to modify a coordinate system, VPG displays the CSYS DEFINITION CARD after the DEFINE COORDINATE CARD LIST allowing the user to key in the exact XYZ coordinates of the system.

-DEFINE CARDS
CURVE

This function defines a load curve.

1. When curve is selected VPG displays the following options:
   CREATE
   - When CREATE is selected, VPG prompts:

   > ENTER DATA (TIME & VALUE) FOR POINT 1 OR END
   - At least two points must be defined to create a curve.
   - Once the two points are defined, the curve is displayed.
   - Refer to the LS-DYNA User's Manual (Section 9.7) for further description.

   DELETE
   - Displays CURVE CARD LIST and allows user to select and delete curves.

   LIST
   - Displays CURVE CARD LIST.

   MODIFY
   - Displays the CURVE CARD LIST and then the CURVE DEFINITION CARD once a curve has been selected.

READ CURVE

- Displays the VPG FILE MENU window and allows the user to read in a previously saved curve (.cur) file.

SHOW CURVE

- Displays the DEFINE CURVE CARD LIST with all defined curves. By selecting a listed curve and exiting the list the curve will be displayed.

-DEFINE CARDS

SPRING ORIENTATION

The definition card in the SPRING ORIENTATION subsection defines orientation vectors for discrete springs and dampers.

1. When CREATE is selected, VPG prompts:
   > ENTER NEW VECTOR ID. (X) OR E TO EXIT

2. Once the vector ID has been established, VPG prompts:
   > SELECT SPRING ORIENTATION OPTION
   ALONG VECTOR
   VECTOR NORMAL PLANE
   ALONG 2 NODES
   2 NODE NORMAL PLANE
   EXIT
   - After defining the vector it is displayed with its vector number in the display window.

note: Refer to the LS-DYNA User's Manual (Section 9.12) for further description.

-DEFINE CARDS
VECTOR

This function defines a vector using the coordinates of two nodes or points

1. When **CREATE** is selected, VPG prompts:
   > ENTER NEW VECTOR ID. (X) OR E TO EXIT

2. Once the vector ID has been established, VPG prompts:
   > DEFINE TAIL OF VECTOR
   - Nodes, points or XYZ coordinates can be used to define the vector tail.

3. VPG prompts:
   > DEFINE HEAD OF VECTOR
   - After defining the vector head the vector is displayed with its vector number in the display window.

note: Refer to the LS-DYNA User's Manual (Section 9.16) for further description.

DEFORMABLE TO RIGID

The definition cards of DEFORMABLE TO RIGID specify properties of deformable parts. Deformable parts may be switched to rigid at the start of a calculation using the DEFORM TO RIGID card. Part switching may be specified on a restart or it may be performed automatically using the AUTOMATIC card. And the INERTIA card sets inertial properties for deformable parts that will be switched to rigid.

1. When **CREATE** is selected, VPG prompts:

```
>   SELECT OPTION
    DEFORM TO RIGID
    AUTOMATIC
    INERTIA
    EXIT
```

   - The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual (Section 10) for further description.

EQUATION OF STATE

The definition cards of EQUATION OF STATE specify element properties.

1. When **CREATE** is selected, VPG prompts:

```
>   SELECT OPTION
    LINEAR POLYNOMIAL
    JWL
    SACK TUESDAY
    GRUNEISEN
    RATIO POLYNOMIAL
    POLY W/ENRGY LEAK
    IGNITION & GROWTH
    TABULATED COMPACTION
    TABULATED
    PROPELLANT DEFLAG
    PORE COLLAPSE
    EXIT
```

   - The preceding commands yield varying definition cards. Refer to LS-DYNA User's Manual (Section 12) for further description.

HOURGLASS CARD

This function allows the user to define hourglass and bulk viscosity properties.

1. When CREATE is selected, VPG displays one definition card. Refer to the LS-DYNA User's Manual (Section 13) for further description.

INTERFACE CARDS

This function defines interfaces for linking calculations.

1. When CREATE is selected, VPG prompts:

```
>   SELECT OPTION
    COMPONENT NODE
    COMPONENT SEGMENT
    DISCRETE NODE LINK
    DISCRETE SET LINK
    LINK SEGMENT
    LINK EDGE
    JOY
    SPRINGBACK
    EXIT
```

   - All the preceding commands yield varying definition cards except for SPRINGBACK. Refer to the LS-DYNA User's Manual (Section 17) for further description of the above commands.

Continued on next page.

2. Upon selecting SPRINGBACK, VPG displays a pop up window with the following selections:

   <NONE>
   NIKE3D
   DYNA3D
   NASTRAN
   EXIT

3. After entering the analysis type, VPG displays a second pop-up window with the selections:

   <NONE>
   THICKNESS
   NO THICKNESS
   EXIT

   - Once the thickness is determined, VPG displays the appropriate definition card. Refer to the LS-DYNA User's Manual (Section 17) for further description of the above command.

SEATBELT

This function defines a seatbelt.

1. When **SEATBELT** is selected, VPG prompts the following card options:

   > SELECT OPTION
   ACCELEROMETER
   PRETENSIONER
   RETRACTOR
   SENSOR
   SLIPRING

   - Each of these seatbelt functions yield various definition cards. For further description of these cards refer to the LS-DYNA User's Manual, Section 11.

TERMINATION

This function defines the analysis termination point in reference to a particular node or body. The nodal stop condition works on the global coordinate position, while the body stop condition works on the relative global translation.

1. When CREATE is selected, VPG prompts:

   > SELECT OPTION
   BODY
   NODE

   - The preceding commands yield varying definition cards. Refer to the LS-DYNA User's Manual (Section 25) for further description.

SUPERELEMENT

The functions in the SUPERELEMENT (S.E.) menu allow the user to assign and modify SUPERELEMENT I.D. numbers within a VPG database (field 9 of a NASTRAN bulk data GRID card). The following options are available in the SUPERELEMENT menu.

SUPERELEMENT

ASSIGN RESIDUAL
ASSIGN S.E. ID
CHANGE S.E. ID
IDENTIFY NODE/POINT
IDENTIFY S.E. ID
VIEW
EXIT

A detailed description of each function is given in the following sections.

note: The SUPERELEMENT option does not appear in the PRE PROCESSING MENU when the analysis code is LS-DYNA.

ASSIGN RESIDUAL

This function allows the user to assign a residual I.D. number to selected nodes displayed in the VPG database (SUPERELEMENT I.D. = 0).

1. VPG prompts:

```
>   SELECT AN OPTION FOR RESIDUAL:
?   INDEPENDENT NODES
    DEPENDENT NODES
    SELECT NODES
    EXIT
```

   - If the user selects INDEPENDENT NODES, VPG automatically assigns a residual I.D. number to the independent nodes of all the displayed RBE2 and RBAR elements.
   - If the user enters SELECT NODES, VPG assigns a residual I.D. number to the nodes that are specified by the user.

2. When SELECT NODES is entered, VPG prompts:

```
>   SELECT NODES
```

   - The user specifies the nodes that are to receive a residual I.D. number.
   - Once the user selects the residual nodes, DONE or EXIT terminates this function. VPG denotes the residual nodes with a small zero (0) on the screen above the appropriate nodes.

ASSIGN SUPERELEMENT I.D.

This function is a one-command operation that assigns a SUPERELEMENT I.D. to all nodes in the entire VPG database (the SUPERELEMENT I.D. is the same as the PID number of the corresponding part).

CHANGE SUPERELEMENT I.D.

This function allows the user to change or assign a SUPERELEMENT I.D. to selected nodes.

1. VPG prompts:

Continued on next page.

> ENTER NEW SUPEREL. I.D. OR 0(ZERO) TO EXIT

- The user may now enter the desired SUPERELEMENT I.D.

2. VPG prompts:
   > NEW S.E. ID DOES NOT EXIST, O.K. TO CONTINUE (Y/N)
   - YES will prompt step 3.
   - NO will exit this function.

3. VPG prompts:
   > SELECT NODES FOR NEW S.E. ID
   - Once the user selects the S.E. nodes, DONE or EXIT terminates this function. VPG displays the new S.E. ID numbers on the screen above the appropriate nodes.

IDENTIFY NODE/POINT

This function identifies any point/node and its corresponding global location within the X, Y, Z coordinates.

1. VPG prompts:
   > PICK NODES/POINTS
   - If the user selects ALL NODES, VPG displays the numbers of the nodes at their node locations and returns the user to the NODE OPTIONS menu.
   - If the user selects NODE, VPG displays the number of the node nearest to the cursor and echoes the following message on the command line:
     > NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - If the user selects POINT, VPG displays the number of the point that is nearest to the cursor and echoes the following message on the command line:
     > POINT xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - DONE or EXIT returns the user to the SUPERELEMENT OPTIONS menu.

IDENTIFY SUPERELEMENT I.D.

This function allows the user to identify the SUPERELEMENT I.D. number of a selected node(s).

Continued on next page.

1. VPG prompts:

   > SELECT NODES/POINTS

   - ALL NODES displays the SUPERELEMENT ID numbers of the nodes.
   - The user may select nodes randomly with the cursor to display the node and SUPERELEMENT I.D. numbers.
   - DONE or EXIT return the user to the SUPERELEMENT OPTIONS menu.

VPG MODULES

VPG MODULES include the following sections: ROAD MENU, SUSPENSION MENU, and TIRE MODEL. The functions in these menus allow the user to quickly and easily construct components required for proving ground simulations.

ROAD DEFINITION

The ROAD DEFINITION menu allows the user to define and save road surfaces. Models may be created using the modeling functions available in eta/VPG, imported using VPG's IGES translator or selected from VPG's library of road surfaces.

VPG's library currently consists of 11 models. The road surfaces included in this library are digitized models of roads found on the MGA Research, Burlington WI proving ground facility, and make up the majority of the 20,000 mile General Durability Test (see the eta/ VPG Applications Manual, Section 1.5).

By default, the road surfaces are defined as rigid bodies with nominal mass. At the user's discretion, the road surface may be modeled using deformable material definitions.

The following functions are included in VPG's ROAD DEFINITION menu:

ROAD DEFINITION

CREATE ROAD SURFACE
DELETE
MOVE ROAD SURFACE
SHOW ROAD SURFACE
COMBINE
SAVE
SELECT FROM LIBRARY
SUMMARY
EXIT

A detailed description of each function is given in the following sections.

CREATE ROAD SURFACE

This option allows the user to defined a road surface by selecting an existing F.E. model.

1. VPG prompts:
   > ENTER NEW ROAD SURFACE NAME OR "STOP" TO EXIT

2. Once a name has been entered, VPG prompts:
   > ONLY PLATE ELEMENTS ARE ALLOWED IN ROAD SURFACE

DELETE

This option allows the user to delete a defined road surface.

1. VPG prompts:
   > PICK ELEM (C) OR ROAD NAME (R)
   ? CURSOR PICK AT ELEMENT
   DONE
   EXIT
   ROAD NAME

2. Once the road surface is selected, VPG prompts:
   > XXXX ELEMENTS IN PART XXXX OF ROAD XXXX
   - The road is deleted.
   - DONE or EXIT returns the user to the ROAD DEFINITION MENU.

MOVE ROAD SURFACE

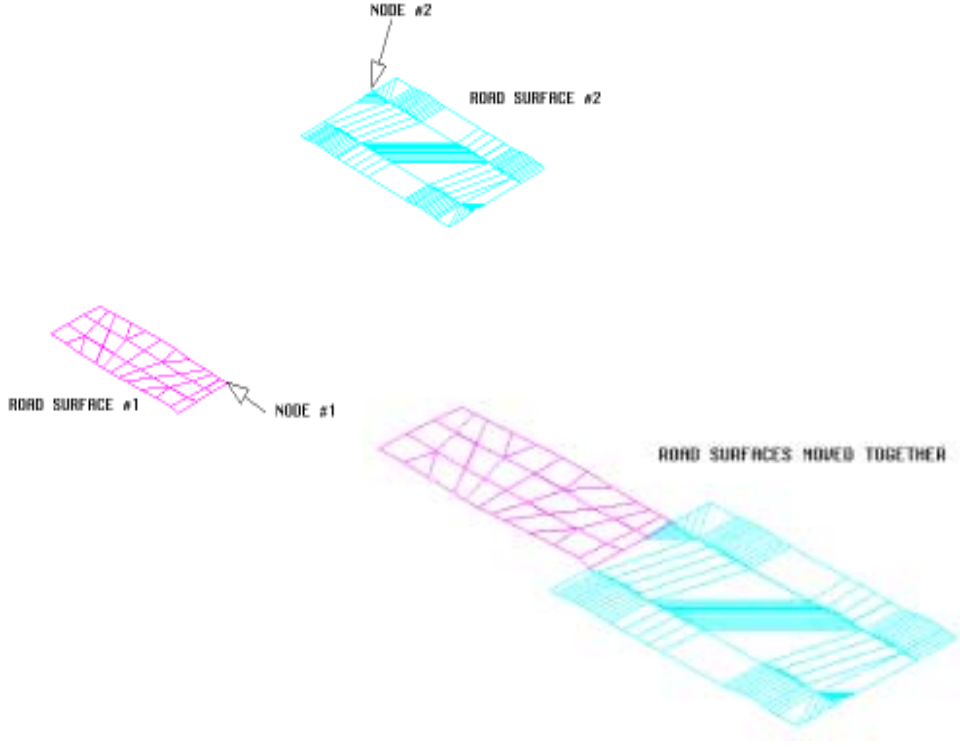
This option allows the user to translate a defined road surface. It is used, primarily, to move two road surfaces together, before combining them into one.

1. VPG prompts:
   > MOVE ROAD SURFACE IN DATABASE
   > SELECT A NODE IN THE ROAD SURFACE TO BE MOVED

2. After the first node is selected, VPG prompts:

   > SELECT A CORRESPONDING NODE FOR THE NEW LOCATION

   - After the second node is selected, the road surface is moved by aligning the first selected node with the second.



SHOW ROAD SURFACE

This option allows the user to highlight a selected road surface.

COMBINE

This options allows the user to combine 2 road surfaces into a single, continuous road surface. It is used in conjunction with the MOVE ROAD SURFACE function.

1. VPG prompts:
   > COMBINE ROAD SURFACES IN DATABASE
   > SELECT AN ELEMENT ON THE FIRST ROAD SURFACE
   • The selected road surface is highlighted in white.

2. VPG prompts:
   > SELECT AN ELEMENT ON THE SECOND ROAD SURFACE
   • The selected road surface is highlighted in white.

note: The selected surfaces must be in contact otherwise VPG will prompt:
> BEFORE COMBINING PLEASE MOVE TWO ROADS TOGETHER

3. VPG prompts:
   > ENTER COMBINED ROAD SURFACE NAME
   • The combined road surface will have a new name.

4. Once the name is entered, VPG prompts:
   > ROAD SURFACES COMBINED SUCCESSFULLY

SAVE

This option allows the user to save a user defined or modified road surface to a user defined road surface library. Multiple road surface libraries can be created with multiple road surfaces in each.

1. VPG prompts:
   > SAVE ROAD SURFACE IN LIBRARY
   > ENTER ROAD LIBRARY FILE NAME OR "STOP" TO EXIT
   • This creates the road surface library file or specifies a previously defined library.

Continued on next page.

2. VPG prompts:
   > ENTER ROAD SURFACE NAME
   - Once the road surface name is entered the road surface is saved to the road surface library.

3. To retrieve a saved road surface, access the USER DEFINED key in the SELECT FROM LIBRARY menu. VPG prompts for the library name. Once the library is specified, VPG lists the road surfaces in the MENU WINDOW.

SELECT FROM LIBRARY

This option allows the user to read a Road Surface from the library, into the VPG database.

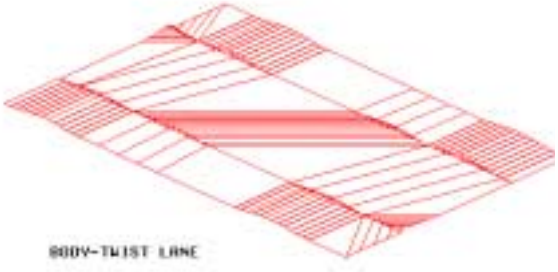
1. VPG prompts:
   > SELECT ROAD SURFACE FROM LIBRARY
   > ENTER COMMAND
   ? USER DEFINED
   - Prompts the user for a previously defined road surface library name and allows the user to select saved road surfaces from that library.

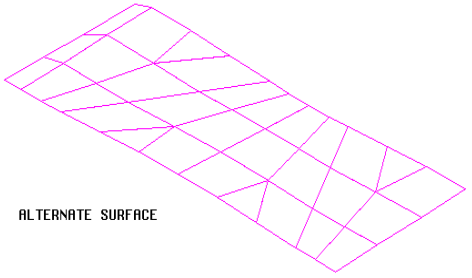
   BODY TWIST LANE
   ALTERNATE SURFACE
   POTHOLE1 TRACK
   POTHHOLE2 TRACK
   RIPPLE1 TRACK
   RIPPLE2 TRACK
   CHATTERSTRIP
   COBBLESTONE TRACK
   WASHBOARD1 SURFACE
   PAVE SURFACE
   WASHBOARD2 SURFACE
   EXIT

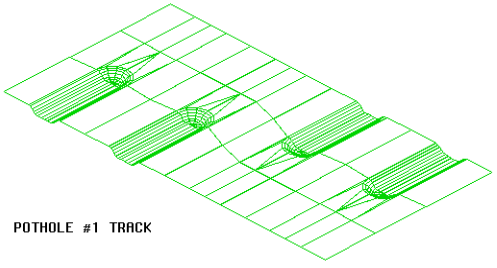
   Road Surface Library Models

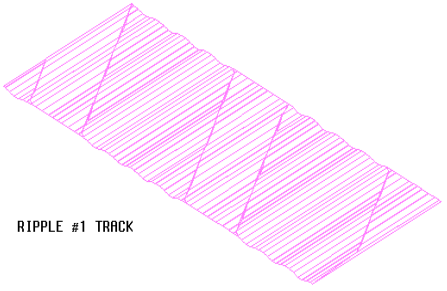
   - The desired road surface is displayed on the screen.

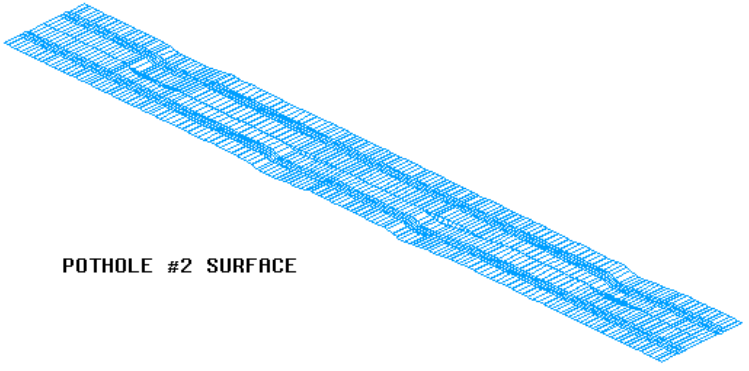
Continued on next page.

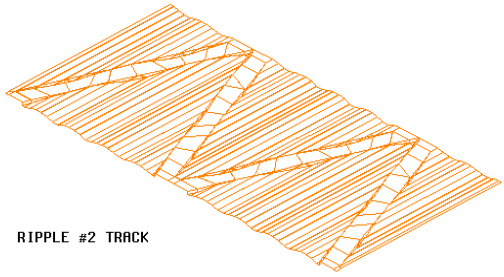

BODY-TWIST LANE

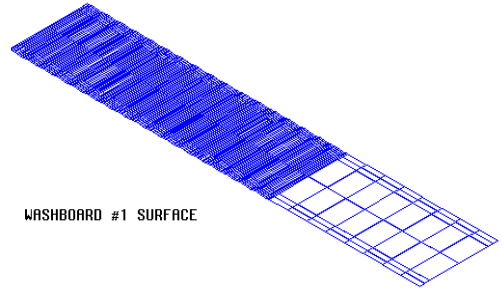

ALTERNATE SURFACE

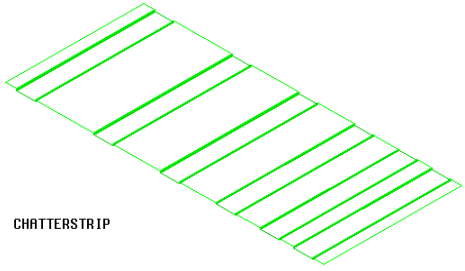

POTHOLE #1 TRACK

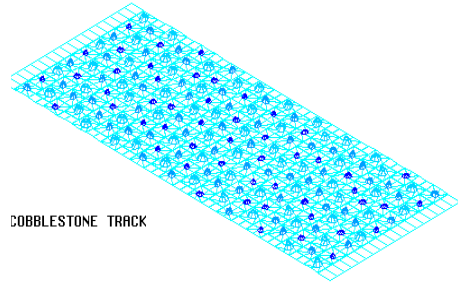

RIPPLE #1 TRACK

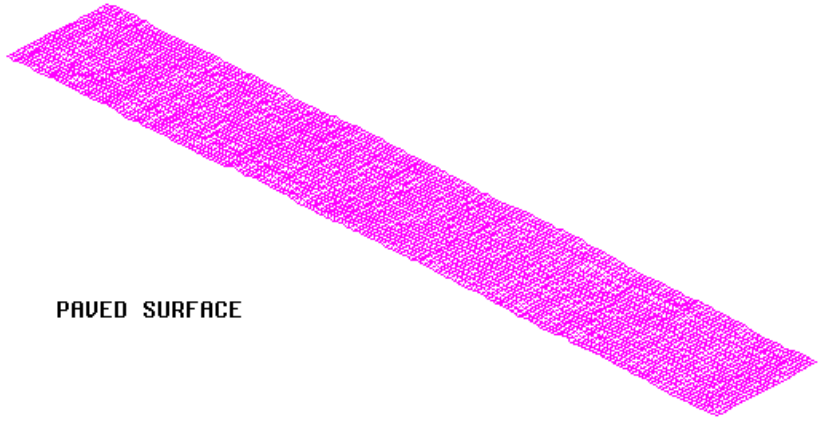

POTHOLE #2 SURFACE

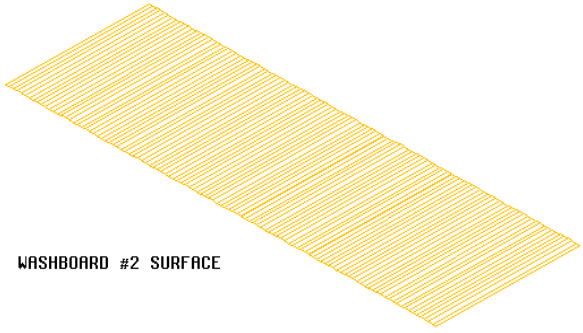

RIPPLE #2 TRACK


WASHBOARD #1 SURFACE


CHATTERSTRIP


COBBLESTONE TRACK

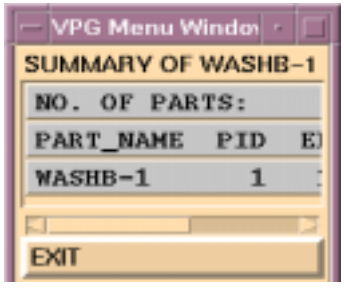

PAVED SURFACE



SUMMARY

This option gives a summary of a selected road surface, i.e.. Part Name, number of parts, PID, and number of elements.

1. VPG prompts:
   > SELECTED ROAD SURFACE SUMMARY
   > SELECT AN ELEMENT ON THE TARGET ROAD SURFACE
   - Once the desired road surface is selected, its characteristics are displayed in the VPG MENU WINDOW.
2. VPG prompts:
   > ENTER CR TO EXIT



SUSPENSION MENU

The functions in the SUSPENSION MENU allow the user to auto-generate Front and Rear Automotive Suspension beam models from a library of suspension types and easily attach them to the vehicle body model.

The beam chassis/suspension model is commonly used in VPG applications. Although a number of different and detailed finite element suspension models can be used, the simulations will take much longer CPU time than they would for simplified beam models. Unless stress/strain history and distribution information are needed for chassis/suspension durability analyses, the beam model is recommended and is both adequate and efficient for most VPG applications such as full vehicle structural durability analysis, road load prediction, full vehicle system NVH analysis, and rideand handling analysis. For more information on suspension definition see the eta/VPG Applications Manual, Section 1.3.

Currently 2 Front and 6 Rear Suspension types can be auto-generated.

SUSPENSION MENU

CREATE
DELETE
MOVE
EXIT

A detailed description of each function is given in the following sections.

CREATE

This option allows the user to define and auto-generate suspension models.

1. VPG prompts:

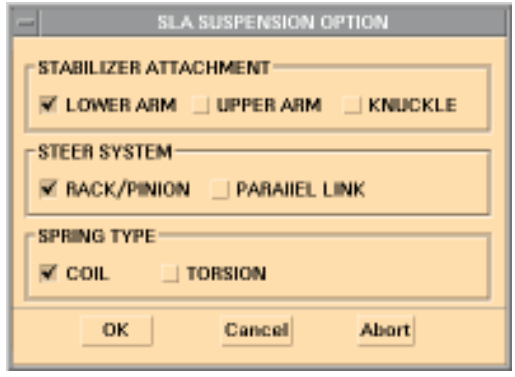
```
>   GENERATE SUSPENSION MODEL
>   ENTER COMMAND
?   FRONT SUSPENSION
    1 SHORT LONG ARM
    2 MCPHERSON A-ARM
 — — — — — — — — — —
    REAR SUSPENSION
    3 MCPHERSON H-ARM
    4 SOLID AXLE
    5 5-LINK
    6 QUADRA LINK
    7 TRAILING ARM
    8 HOTCHKISS
    9 TWIST BEAM
    10 HONDA 5-LINK
    EX-EXIT
```

2. Depending on the suspension type, VPG will display one or both of the following two tables. The first table displays suspension specific options such as stabilizers, springs, and attachments, and is only necesary for some suspension types. VPG prompts:

```
>   MODIFY SUSPENSION OPTIONS
```

   - When the user has finished altering the data and has selected OK, VPG forwards the user to the next suspension table on the following page.

Continued on next page.



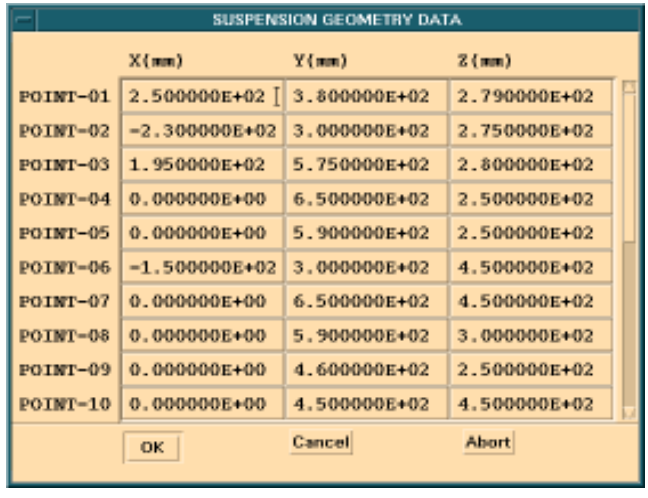
3. VPG prompts:

   > MODIFY GEOMETRY DATA

   - When the user has finished altering the data and has selected OK, VPG prompts:

   > ACCEPT THESE GEOMETRY DATA? (Y/N)

   - YES forwards the user to the next suspension table (step 4).
   - NO returns the user to the SUSPENSION GEOMETRY table.

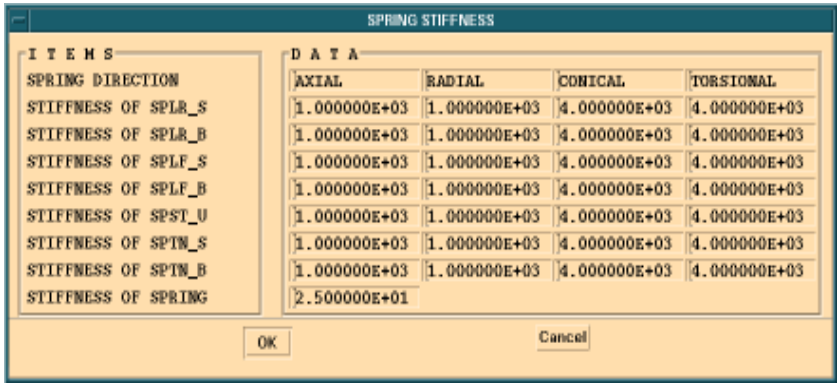


Continued on next page.

4. VPG displays the following pop-up window with suspension stiffness data, and VPG prompts:
   > MODIFY STIFFNESS DATA
   > SELECT AN ITEM TO EDIT
   - When the user has finished altering the data and has selected OK, VPG prompts:

   > ACCEPT STIFFNESS DATA? (Y/N)
   - YES forwards the user to the next suspension table (step 5).
   - NO returns the user to the SUSPENSION STIFFNESS table.

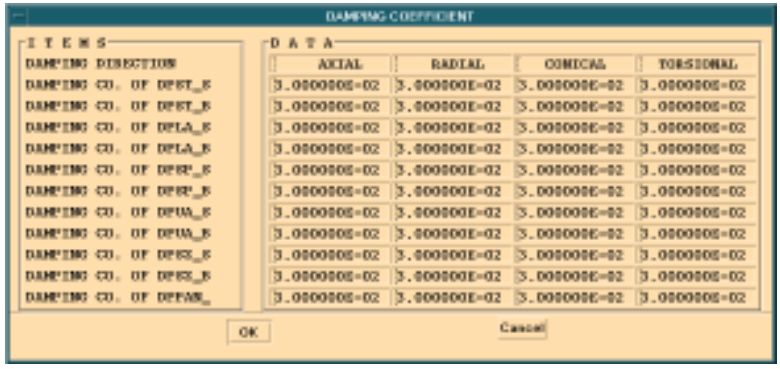
note: If the user selects cancel at any time during the suspsension creation process the process is ended and the suspension is not defined.



5. VPG displays the following pop-up window with damping coefficient data, and VPG prompts:
   > MODIFY DAMPING COEFFICIENT DATA
   > SELECT AN ITEM TO EDIT
   - When the user has finished altering the data and has selected OK, VPG prompts:

   > ACCEPT DAMPING COEFFICIENT DATA? (Y/N)
   - YES forwards the user to the next suspension table (step 6).
   - NO returns the user to the DAMPING COEFFICIENT table.

Continued on next page.



6. VPG displays a pop-up window with node coordinate data. The extra nodes are used to define spring, damping and axle direction. VPG prompts:

   > MODIFY EXTRA NODE COORDINATES

   > SELECT AN ITEM TO EDIT

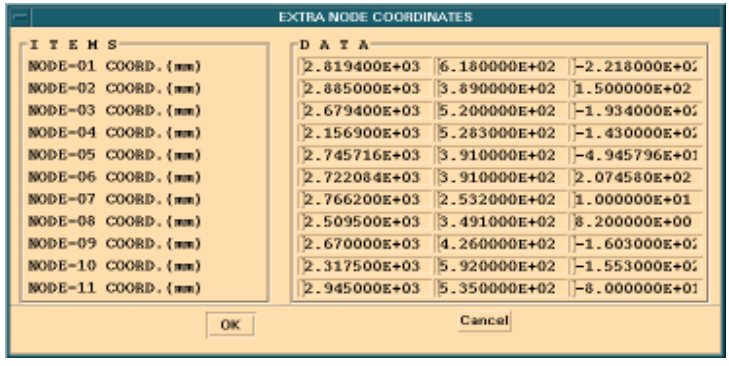
   - When the user has finished altering the data and has selected OK, VPG prompts:

   > ACCEPT EXTRA NODE COORDINATES? (Y/N)

   - NO returns the user to the extra NODE COORDINATES table.
   - If YES is entered VPG prompts:

   > USE DEFAULT MASS, CG. & INERTIA MOMENT? (Y/N)

   - YES completes the suspension.



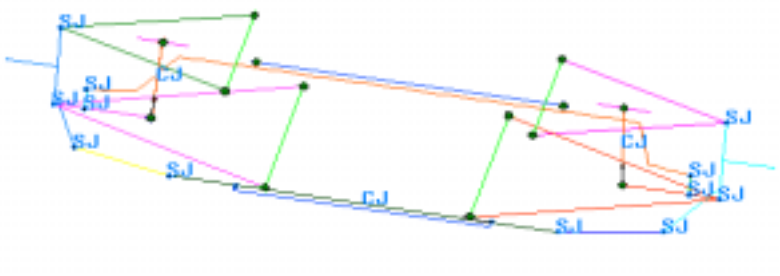
7. If NO is entered VPG prompts the user to select the parts to modify:

Continued on next page.

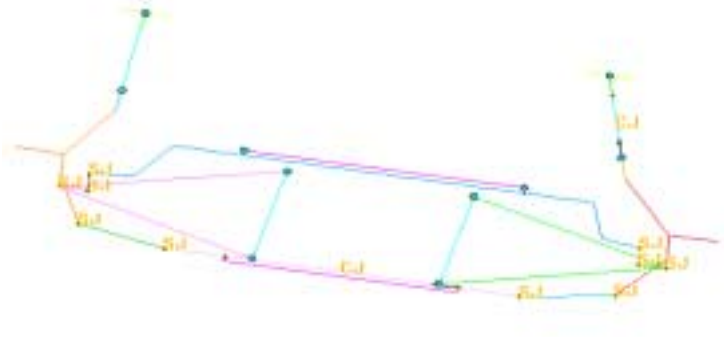
```
>   PICK PARTS TO MODIFY
    [list of all parts]
```

8. Once the parts are selected and the user has exited the list VPG prompts the user:
   ```
   >   MODIFY MASS A CG. CENTER
   >   MODIFY INERTIA MOMENT
   ```
   - VPG displays a pop-up window with the parts' properties allowing the user to alter them.

Front Suspension Library

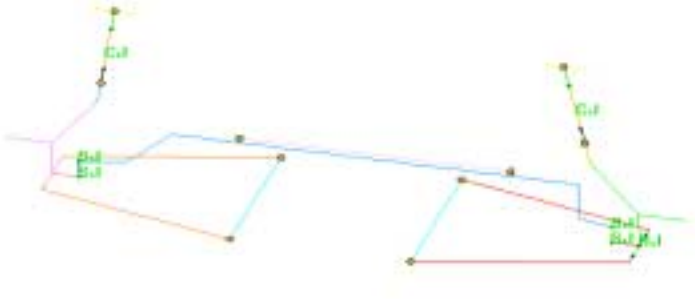


SHORT-LONG A-ARM SUSPENSION

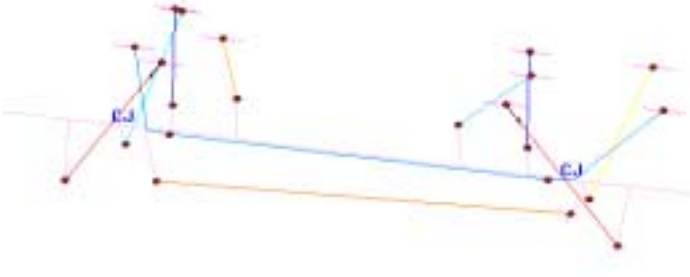


MCPHERSON A-ARM FRONT SUSPENSION

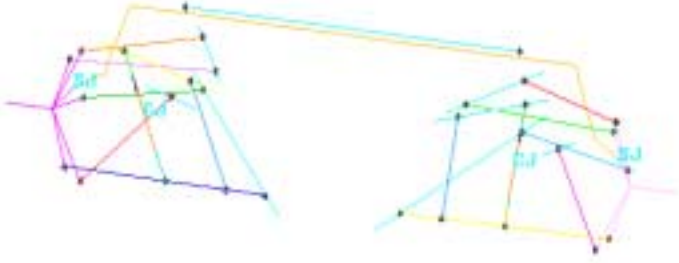
Rear Suspension Library



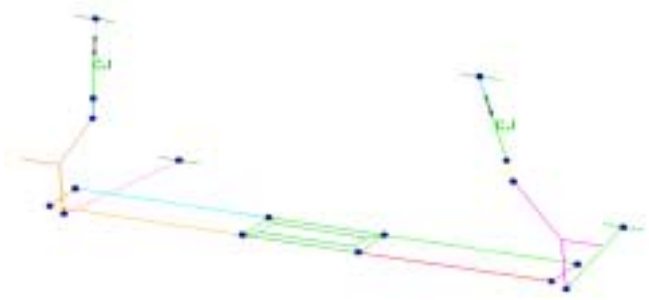
MCPHERSON H-ARM REAR SUSPENSION



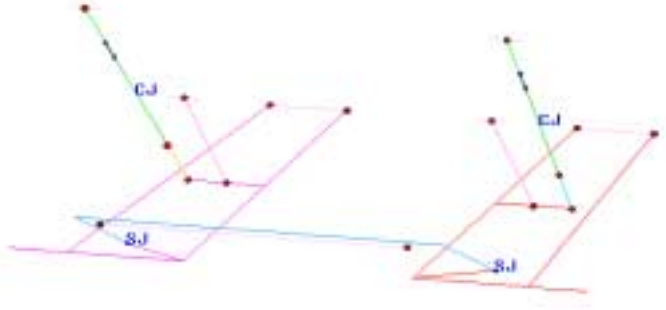
TWIST BEAM REAR SUSPENSION



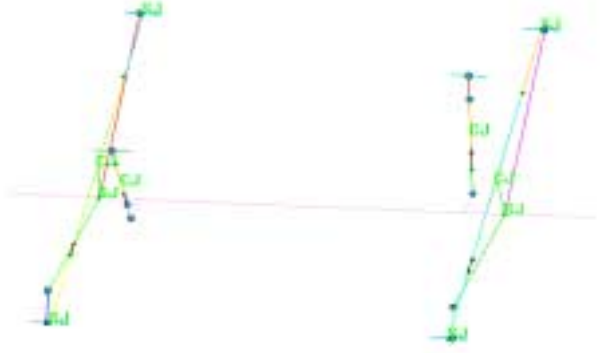
5 -LINK REAR SUSPENSION



QUADRA LINK REAR SUSPENSION



TRAILING ARM REAR SUSPENSION



HOTCHKISS REAR SUSPENSION

DELETE

This option allows the user to delete defined suspension models from the VPG database.

1. VPG prompts:
   > PICK AN ELEMENT TO DELETE SUSPENSION
   - Once an element from the suspension is selected, VPG prompts:

   > OK TO DELETE THIS SUSPENSION? (Y/N)
   - YES deletes the suspension model and returns the user to the SUSPENSION MENU.
   - NO returns the user to the SUSPENSION MENU.

MOVE

This option allows the user to translate auto-generated suspension models. Its purpose is to allow the user to quickly align the auto-generated suspension model with the F.E. body model for attachment.

1. VPG prompts:
   > PICK AN ELEMENT TO TRANSLATE SUSPENSION
   ? PICK TWO NODES
   INCREMENTAL XYZ
   UNDO LAST
   EXIT
   - Once the two locations have been determined the suspension is moved with the second coordinate now occupying the first.

TIRE MENU

The tire model is one of the most important components in a VPG analysis. Although modern pneumatic tires have very complex structures, the VPG/tire model concentrates only on the most important global mechanical properties of the tire. These properties ensure the proper transfer of forces and displacements from the tire/road contact interface to the vehicle chassis/suspension and body structure.

A key advantage of this tire model is its simplicity, giving CPU economy. The tire has the ability to interact with three dimensional road surfaces in a manner consistent with actual tires. This allows for the prediction of lateral and longitudinal force within the vehicle suspension with greater accuracy.

VPG's tire generator creates tires with the following characteristics and properties:

The tire tread and chafer are Mooney-Rivlin type rubber materials, modeled with eight-node solid elements. The tire top ply and side wall are modeled as shell elements with elastic material. The wheel is made of rigid shell elements. A “bottom-up stopper”, made of a layer of elastic shell elements, is connected to the rim. A contact interface is defined between this layer and the tire top-ply. Once the tire deformation is large enough, this interface will provide extra stiffness to the tire. The stopper’s height and material property can be adjusted to achieve the required secondary stiffness.

Tire inflation pressure is defined according to the formula: $\boldsymbol{pV/T}$ **= constant** where p is pressure; V is the volume confined within the top ply, side wall, and the wheel hub; and T is a temperature assumed to be constant during the simulated testing process.

The finished VPG/tire model contains a total of 2441 nodes and 2264 elements (including 1304 shell elements and 960 solid elements).

The TIRE MENU allows the user to auto-generate tire models with user defined characteristics.

TIRE MENU

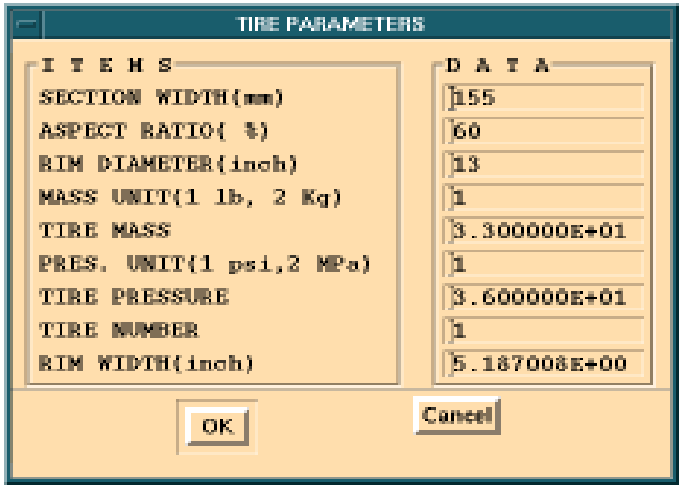
GENERATE TIRE MODEL
DELETE TIRE MODEL
COPY TIRE MODEL
EXIT

A detailed description of each function is given in the following sections.

GENERATE TIRE MODEL

This option allows the user to auto-generate multiple tire models based on user defined parameters.

1. VPG displays a pop-up window with tire parameter data and prompts:
   > DEFINE/MODIFY TIRE DATA
   > SELECT AN ITEM TO EDIT



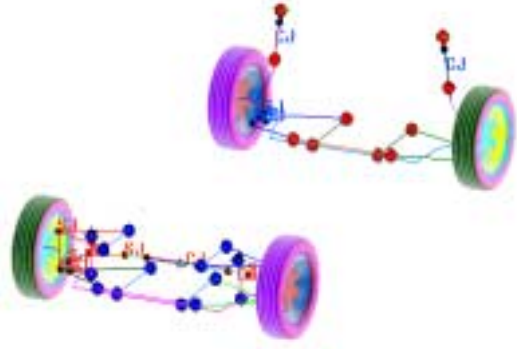
2. When the user has exited the tire parameter window, VPG displays the tire cross-section along with mass and pressure values and prompts:
   > PROCESSING ELEMENT AND NODAL DATA
   > DO YOU ACCEPT THIS TIRE PROFILE? (Y/N/ABORT)
   - YES returns the user to the tire parameter window.

Continued on next page.

3. If the user accepts the tire profile, VPG prompts:
   > DEFINE WHEEL CENTER LOCATION FOR TIRES

4. Once the tire location is defined, VPG displays the tire material density and tire model:
   > THE MATERIAL XXXXXXX DENSITY IS XXXXXXXX

note: for tire tuning procedures see the eta/VPG applications manual sections 1.6 and 2.4



DELETE TIRE MODEL

This option allows the user to delete the currently defined tire models.

1. VPG prompts:
   > DELETE TIRE BY ELEMENT PICK

2. Once the tire is selected, it is highlighted and VPG prompts:
   > DO YOU WANT TO DELETE THIS TIRE? (Y/N/ABORT)
   - If YES is selected, the tire will be deleted.

COPY TIRE MODEL

This function allows the user to copy a tire model and its characteristics to a specific location.

1. VPG prompts:
   > CHOOSE A TIRE FOR COPYING BY PICKING AN ELEMENT

2. After selecting the tire, VPG prompts:
   > DO YOU WANT TO CHOOSE THIS TIRE? (Y/N/ABORT)

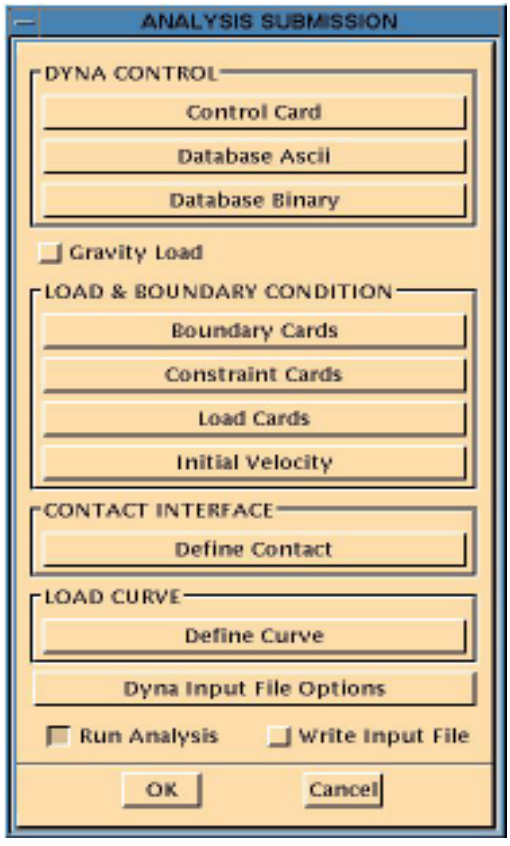
3. The user is then prompted for the location of the new tire. Once entered, the tire is copied to that location.

ANALYSIS EXECUTION

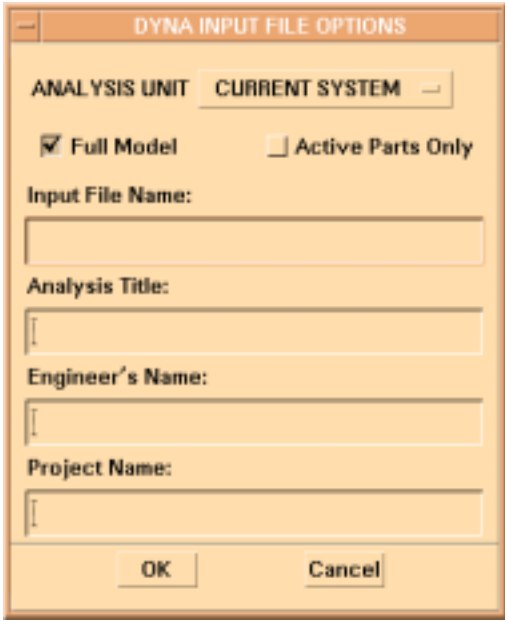
VPG allows the user to submit an analysis for execution directly from the VPG interface using the functions found in the RUN ANALYSIS menu. If running an analysis from outside the VPG processor, the user should refer to the introduction of the *LS-DYNA Keyword User's Manual.*

RUN ANALYSIS

This option allows the user to interactively run the VPG solver, LS-DYNA, from the Graphical User Interface window. The option buttons in the ANALYSIS SUBMISSION window (CONTROL CARD, BOUNDARY CARDS, CONSTRAINT CARDS, etc.) provide access to the neccessary functions for completing a DYNA deck. The toggle button at the bottom of the window, RUN ANALYSIS, submits the job for analysis and writes an input file when the user selects OK. The WRITE INPUT FILE toggle writes an input file without submitting an analysis.



The DYNA INPUT FILE OPTIONS button displays a window (see figure below) allowing the user to set the analysis unit for the job, the input file name, the analysis title, then engineer's name submitting the analysis, the project name and the parts to be included.



RESTART

The RESTART function in the main menu allows the user to switch from the current VPG database to a different VPG database in either the same directory (default) or in a user specified directory. Before restarting with a new database, VPG reminds the user if the current database has been modified and allows the user the option to save the database before restarting.

POST PROCESSING

VPG's post processing functions include a fatigue analysis program, a signal processing program, and a wide variety of functions which allow the user to graphically display and manipulate solver code result files and visualize the results of an analysis with XY plots.

FATIGUE EVALUATION

VPG's fatigue evaluation program evaluates fatigue life for both LS-DYNA and NASTRAN analysis. Before executing a fatigue analysis, the FATIGUE OPTION function should be accessed to control the input data for the analysis. Due to the lengthy run time of fatigue analyses, VPG allows the user to return to modeling while the job is running.

To access the fatigue analysis result file (**lifecyc.pac**), the user should first read in the original model file using either the READ DYNA or READ NASTRAN BULK functions (depending on the analysis type) in the FILE MANAGER menu. The user should then enter the POST PROCESSING menu and load the **lifecyc.pac** file via the LOAD RESULT FILE window.

Once the **lifecyc.pac** file is loaded the CONTOUR FILL function is used to post process the results. There is only one step available when post processing a fatigue result file. The variables displayed in the SELECT VARIABLE window such as MAX VON MISES, SIGYY, SIGXX, have no bearing on the interpretation of the result file. All variables display the same. The default color setting for the CONTOUR FILL function sets blue at the minimum end of the scale and red at the maximum. For this reason reversing the color mode using the REVERSE CONTOUR in DISPLAY OPTIONS is more meaningful for reading the fatigue data (red=weakest, blue=strongest).

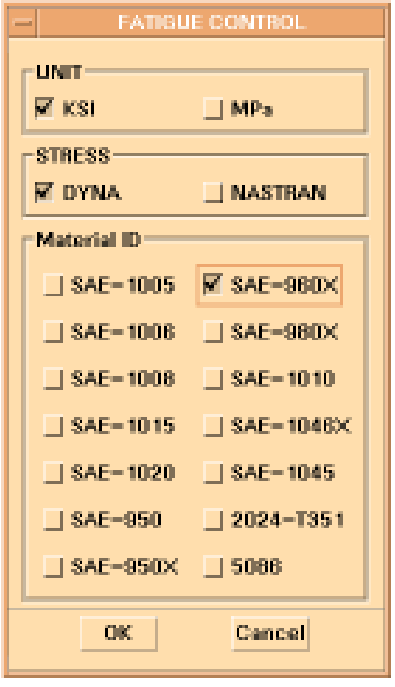
The FATIGUE menu consists of two options:

FATIGUE OPTION
EXECUTE FATIGUE

These functions are described in detail on the following pages.

FATIGUE OPTION

The FATIGUE CONTROL window determines the unit, analysis type and type of material used in the model for the fatigue evaluation. The user should set the fatigue control to the correct parameters before executing the fatigue analysis.



Continued on next page.

EXECUTE FATIGUE

The EXECUTE FATIGUE function allows the user to submit the fatigue analysis and return to VPG to continue modeling. After analysis submission VPG prompts:

```
>   FATIGUE ANALYSIS RUNNING IN THE BACKGROUND
```

- When fatigue analysis is complete, a packed NASTRAN result file is written (**lifecyc.pac**) for post processing.

Continued on next page.

POST PROCESSING

The functions in the POST PROCESSING menu allow the user to graphically display and manipulate solver code result files with contour and deformation animation, contour plots, and geometry deformation.

POST PROCESSING MENU

ANIMATE CONTOUR
ANIMATE DEFORMATION
CONTOUR FILL
CONTOUR LINES
DEFORMED SHAPE
ELEMENT STRESS
READ RESULT FILE
SUMMARY
VIEW
EXIT

A detailed description for each function is given in the following sections.

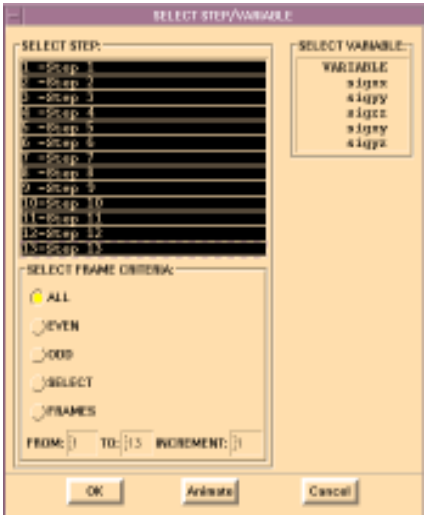
When entering the POST PROCESSING menu, if the database is empty, VPG automatically prompts the user to read a binary (**.pp**) or an LS-DYNA plot file (**d3plot**) into the database. If a **d3plot** file is selected, the user is prompted to select the LS-DYNA version that created the file.

note: It is recommended an empty database be used to post-process the **.pp** and **d3plot** files since these files already contain the undeformed model.

ANIMATE CONTOUR

The functions in this section animate transient analysis stress contours in real time.

1. VPG displays the SELECT STEP/VARIABLE window.
   - A variable must be selected to animate.



- Selecting OK forwards the user to the CONTOUR ANIMATION submenu (see PRE-ANIMATION this section).
- Selecting ANIMATE begins the animation and displays the STATIC ANIMATION submenu (see POST ANIMATION this section).

ANIMATE DEFORMATION

This function animates the displacement of transient and static analysis in real time. Transient animation occurs automatically, whereas non-transient (static, linear punch file) animation requires the user to specify some parameters.

1. VPG displays the SELECT STEP window (see ANIMATE CONTOUR).
   - Selecting OK forwards the user to the DEFORMED SHAPE submenu (see

Continued on next page.

PRE-ANIMATION this section).

- Selecting ANIMATE begins the animation and displays the STATIC ANIMATION submenu (see POST ANIMATION this section).

CONTOUR FILL

This function superimposes a fill-color contour image onto the current model. The color legend, located in the upper right hand corner of the screen, displays the corresponding contour values. The location of the highest contour values on the model is labeled with an asterisk (*), and the lowest location is labeled with a zero (0).

1. VPG displays the SELECT STEP/VARIABLE window (see ANIMATE CONTOUR).
   - Selecting OK or PLOT plots the step and forwards the user to the CONTOUR PLOT submenu (see POST ANIMATION this section).

note: If the result is based on the elements (not nodes), the contour plot may not show the absolute maximum and miminum value of the result. The reason is that the contour value at each node is obtained by averaging the centroidal result from the elements connecting to the node. In this case, the ELEMENT STRESS function will show the absolute results from the analysis.

CONTOUR LINES

This function superimposes a color, wire-frame, contour-line plot of the component result onto the current model. The color legend, located in the upper right hand corner of the screen, displays the corresponding contour values. The location of the highest contour value in the model is labeled with an asterisk (*), and the lowest location is labeled with a zero (0).

1. VPG displays the SELECT STEP/VARIABLE window (see ANIMATE CONTOUR).
   - Selecting OK or PLOT plots the step and forwards the user to the CONTOUR PLOT submenu (see POST ANIMATION this section).

DEFORMED SHAPE

This function displays the displacement results of the analyzed model. The undeformed shape is displayed in white by default (see UNDEFORMED ON/OFF for more details).

1. VPG displays the SELECT STEP/VARIABLE window (see ANIMATE CONTOUR).
   - Selecting OK or PLOT plots the step and forwards the user to the DEFORMED SHAPE submenu (see POST ANIMATION this section).

ELEMENT STRESS

This function displays the stress results of each individual element as opposed to the contour plots which display the stress results in terms of the average stress at each node.

1. VPG displays the SELECT STEP/VARIABLE window (see ANIMATE CONTOUR).
   - Selecting OK or PLOT plots the step and forwards the user to the CONTOUR PLOT submenu (see POST ANIMATION this section).

READ RESULT FILE

This function reads and processes result files from a variety of different analysis codes.

1. VPG prompts the user to pick a file type. The five available types are:

```
>   ENTER RESULT FILE TYPE
?   NASTRAN PUNCH
    PACKED NASTRAN
    HISTORY
    D3PLOT
    .PP FILE
    DYNA/DEFGEO FILE
    FFT CURVE FILE
    NODOUT TO .PP FILE
```

2. The five options describe distinct file types. (The following chart lists the software that produces each of the files.)

Continued on next page.

3. After the user selects one of the options, VPG displays the VPG FILE MENU (See section 2.11) and prompts:
   > ENTER DATABASE FILE NAME OR "STOP" TO EXIT

4. Once a file is selected, VPG loads the file and returns the user to the POST PROCESSING menu.

NAME	EXTENSION	SOFTWARE TYPE	DESCRIPTION
Nastran Punch	*.pch	NASTRAN	NASTRAN punch file
Packed Nastran	*.pac	NASTRAN	NASTRAN punch file without stress components
History	*.his	LS-DYNA	Transient Analysis Animation File
Dyna3d Plot File		d3plot, etc.	LS-DYNA Transient
Post Processing	*.pp	VPG	Binary File
Deformed Geometry	defgeo	LS-DYNA	Deformation Animation
FFT	*.fre	VPG	Fast Fourier Transform
DYNA NODOUT		DYNA	Time Frequency Domain Conversion

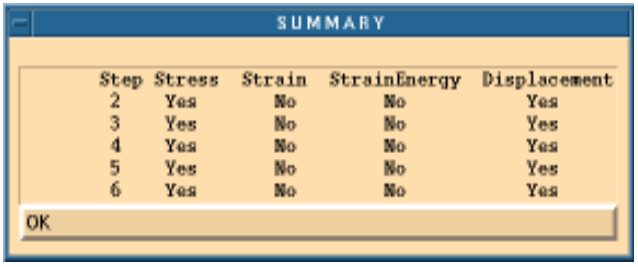
note: VPG automatically creates a Binary Result file (**`filename.pp`**) from the ASCII result file.

note: VPG also reads LS-DYNA plot files with adaptive mesh.

SUMMARY

This function pops up a window that indicates the status of the result file by displaying the stress, strain energy, and the availability of displacement at given time steps.

Continued on next page.


SUMMARY
Step Stress Strain StrainEnergy Displacement
2 Yes No No Yes
3 Yes No No Yes
4 Yes No No Yes
5 Yes No No Yes
6 Yes No No Yes
OK

This window displays the status of the result file.

PRE-ANIMATION

The following functions are found in the CONTOUR ANIMATION and DEFORMED SHAPE submenus. Following is a general description of these commands.

CONTOUR MODE
DISPLAY OPTIONS
SELECT STEP/COMP.(VAR.)
LIST DISPLACEMENT
LIST CONTOUR VALUE
SECTION CUT
ANIMATE
EXIT

CONTOUR MODE

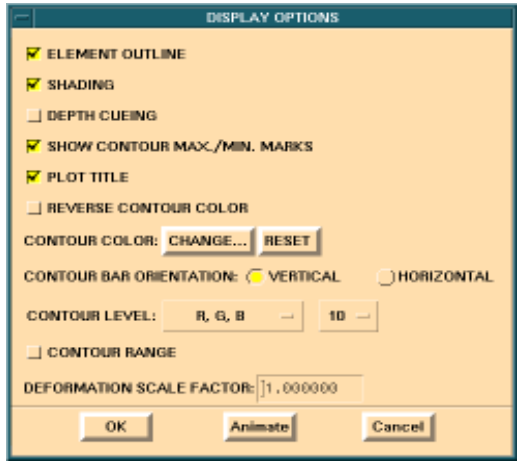
The options in the CONTOUR MODE setting select the STRESS/STRAIN/STRAIN ENERGY/DISPLACEMENT component for the animation.

DISPLAY OPTIONS

The DISPLAY OPTIONS function displays a pop-up window allowing the user to set the viewing parameters for the animation.

Continued on next page.

DISPLAY OPTIONS
window for stress contour.



-DISPLAY OPTIONS

ELEMENT OUTLINE

Turns the element outline ON/OFF.

-DISPLAY OPTIONS

SHADING

This command displays the elements as if they were illuminated by a light source. Elements that are not directly exposed to the light source are appropriately “shaded” to imitate actual shading.

note: Shading with continuous stress levels offers the highest quality of color display.

-DISPLAY OPTIONS

DEPTH CUEING

The DEPTH CUEING function simulates depth perspective, a phenomenon that causes distant objects to appear dimmer than closer objects. When the user activates the DEPTH CUEING command, VPG displays the parts of the model that are intended to appear farther from the viewer at a lower intensity.

-DISPLAY OPTIONS

SHOW CONTOUR MAX/ MIN MARKS

Toggles between the maximum and minimum display markers.

-DISPLAY OPTIONS

PLOT TITLE

The title of the result file is displayed in the top left corner of the display window. This option allows the user to toggle the title on or off.

-DISPLAY OPTIONS

REVERSE CONTOUR COLOR

Reverses the contour color scale.

-DISPLAY OPTIONS

CONTOUR COLOR

CHANGE allows the user to arbitrarily select and change colors for the color bar. RESET returns the contour color bar to its original setting.

-DISPLAY OPTIONS

CONTOUR BAR ORIENTATION

Allows the user to set the CONTOUR COLOR BAR orientation either vertically or horizontally on the screen.

-DISPLAY OPTIONS

CONTOUR LEVEL

Allows the user to set the color scheme (RG, RGB, RGBM, GRAYSCALE and CONTINUOUS) and the number of colors/shades used from 10 to continuous (256).

-DISPLAY OPTIONS

REFERENCE NODE

Allows the user to select a node to reference for displacement and the axes to be displayed.

-DISPLAY OPTIONS

CONTOUR RANGE

Displays two slide bars to set the lower and upper contour range.

-DISPLAY OPTIONS

DEFORMATION SCALE FACTOR

This command adjusts the default scale factor of the deformed shape plot. The default scale is the maximum deflection scaled to 7.5% of the largest dimension of the screen size. For transient type animation, the time scale (1.00) is used as default. VPG still allows the user to adjust the scale factors of either the deformed shape plot or the deformed shape animation.

LIST DISPLACEMENT

This function lists displacement values for selected nodes in the X, Y, and Z planes.

LIST CONTOUR VALUE

This function lists the contour values of selected elements or nodes.

SELECT STEP/COMP. (/VAR.)

The SELECT STEP/COMP. function displays the SELECT STEP VARIABLE window allowing the user to set the number of steps in the animation. See ANIMATE CONTOUR of this section for a picture of this window and the specific components to be displayed.

SECTION CUT

This function cuts a plane to expose the inside of the model during the animation operation. The cutting plane defined is the UV plane.

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/PT FOR ORIGIN
   - VPG is now asking for a coordinate system to create section lines. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Local Coordinate System, Section 3.5.*

2. Once the desired coordinate system is acquired, it will be displayed on the screen.

3. If the user accepts the coordinate system, VPG prompts:
   > DISTANCE FROM PLANE, W=?
   - Users may now input a distance along the W-axis from origin of the selected coordinate system.
   - Any real value is a valid response.
   - The section line will be created at the distance W in the UV plane.

4. VPG prompts:
   > CHANGE SECTION OPTIONS
   FIXED/MOVE SECTION (toggle)
   - This command defines a fixed or moving section plane with respect to the model.

Continued on next page.

NEGATIVE W-AXIS

- The section cut will display the model on the negative w-axis.

POSITIVE W-AXIS

- The section cut will display the model on the positive w-axis.

RE-SCALE CONTOUR LINES

- The default uses the original max. and min. values for contour.

SECTION CUT ONLY

- Displays just the cross section for animating.

SECTION PLUS CURVE
EXIT

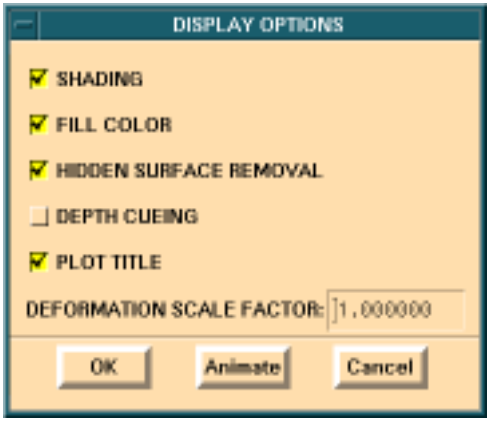
POST ANIMATION

The following functions are found in the STATIC ANIMATION, DEFORMED SHAPE, and CONTOUR PLOT submenus. Following is a general description of these commands.

DISPLAY OPTIONS
ELEMENT OUTLINE ON/OFF
INDIVIDUAL FRAME
FASTER SPEED
SLOWER SPEED
MOVE LIGHT SOURCE
PART ON/OFF
RECALL LAST VIEW
SAVE MPEG
EXIT

DISPLAY OPTIONS

The DISPLAY OPTIONS function displays a pop-up window allowing the user to set the viewing parameters for the animation.



Display options window for deformed shape plot

ELEMENT OUTLINE ON/OFF

Allows the user to toggle off the white outline of the displayed elements.

INDIVIDUAL FRAME

This offers the user a submenu to select a specific frame of animation to view or inquire on nodes elements, particular stress, strain or displacement at that particular time step.

SELECT STEP/COMP.

The SELECT STEP/COMP. function displays the SELECT STEP VARIABLE window allowing the user to set the number of steps and desired component in the animation. See ANIMATE CONTOUR of this section for a picture of this window.

FASTER SPEED

This function allows the user to increase the animation speed after using the SLOWER SPEED command. Faster is the default setting.

SLOWER SPEED

This function allows the user to decrease the animation speed.

MOVE LIGHT SOURCE

This function allows the user to change the direction of the light source when the model is shaded.

PART ON/OFF

This function allows the user to turn different parts in the model on and off. The part names are displayed in the VPG MENU WINDOW. Selected parts are highlighted in white, when the user exits the window the selected parts will no longer be displayed.

SAVE MPEG

This function prompts VPG to generate an MPEG (.mpg) file of the animation.

GRAPH

The functions in the GRAPH menu enable the user to visualize the results of an analysis with XY plots. VPG offers a wide range of tools to manipulate the information on the display screen with labels, colors, multiple graphs, and a host of advanced filtering techniques e.g., FIR, Butterworth, scaling, smoothing, and averaging.

note: VPG automatically forwards the user into the READ DATABASE command before executing these functions.

GRAPH

ADD CURVE TO GRAPH
ADD MODEL
CLEANUP MEMORY
CURVE OPERATIONS
CLEAR SCREEN
DELETE CURVES
HARDCOPY
OPTIONS
PLOT CURRENT GRAPH
PLOT MULTI GRAPHS
QUERY
RANGE
READ DATABASE
SAVE CURVES
SET DEFAULTS
SELECT GRAPH
TIME<-->FREQUENCY
EXIT

A detailed description of the commands in the GRAPH menu is given in the following sections.

ADD CURVE TO GRAPH

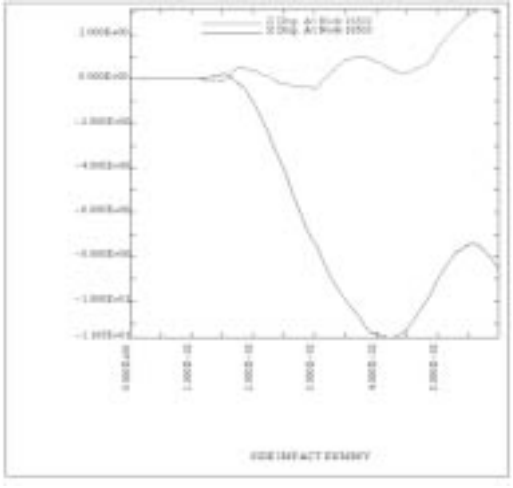
This function plots a selected curve in a selected graph.

1. VPG prompts:
   > SELECT A GRAPH
   ? [list of graphs]
   CURRENT GRAPH

2. VPG prompts:
   > SELECT CURVES FROM MENU TO ADD TO CURRENT GRAPH
   ? [list of curves]
   UNDO
   ABORT
   DONE
   - The user may select a graph from the menu or pick a graph through the multiple graph function.

3. After the user selects the graph, the ADD CURVES MENU lists the curves that are not present in the selected graph. The user can add multiple curves to the active graph.

note: Each curve may be active in only one graph at a time. To display a curve more than once in a single session, the user must enter CURVE OPERATIONS and select the COPY CURVES command in order to retain the two identical curves in resident memory.



CLEANUP MEMORY

This function removes all information regarding the graphs from the screen and memory. This is useful if the user wants to generate a new set of graphs without exiting the GRAPH menu. Exiting the GRAPH menu automatically eliminates the information stored in memory.

1. VPG prompts:
   > CLEANUP MEMORY & SCREEN (YES/NO)?
   ? YES
   NO

2. If the user selects YES, then VPG removes all graph data from memory. Otherwise, VPG returns the user to the GRAPH menu.

CURVE OPERATIONS

This function activates the **CURVE OPERATIONS** tools for manipulating plots.

note: The user may select multiple curves via the menu or mouse.

1. VPG displays:
   ? + (add)
   - (subtract)
   *** (multiply)**
   / (divide)
   - These first four options manipulate the Y-axis values. If the curves have different domains, VPG automatically extends the domains.

   NEGATE
   - This multiplies the Y-axis values by -1 (negative one).

   COPY CURVE
   - This duplicates a selected curve.

   DIFFERENTIATE
   - This superimposes the derivative of the curve on the graph (plots the value of the slope of the curve's tangent).

Continued on next page.

INTEGRATE

- This superimposes the integral of the curve on the graph (plots the value of the area under the curve from start to x).

TRANSLATE

- This moves curves right/left (positive and negative X) and up/down (positive and negative Y).
- The user selects a curve with the mouse.

> ENTER X TRANSLATION OR RETURN NOT TO TRANSLATE X

> ENTER Y TRANSLATION OR RETURN NOT TO TRANSLATE Y

- VPG displays translated curves.

SWITCH AXIS

- Switches the XY-axis for the selected curves.

> SELECT A CURVE (LEFT: SELECT, MIDDLE: DESELECT, RIGHT: DONE)

? SELECT CURVES
[LIST OF CURVES]
UNDO
ABORT
DONE

- The user selects the curves, then enters DONE to execute the command.

SCALE

- This command scales the values of the curves. The user may select the curves from the menu or pick multiple curves with the mouse.

> ENTER X SCALE FACTOR, OR RETURN TO KEEP X SCALE

> ENTER Y SCALE FACTOR OR RETURN TO KEEP Y SCALE

- After the user enters the Y scale, VPG displays the changes.

SMOOTH

- VPG displays the SMOOTH CURVES menu:

? AVERAGING

- This command averages the value of a point with a number of surrounding points (user definable). This "smooths" the curve.

> SELECT CURVES TO BE AVERAGED

> SELECT A CURVE (LEFT: SELECT, MIDDLE: DESELECT, RIGHT: DONE)

- After the user picks a displayed curve from the menu or by

Continued on next page.

mouse, VPG prompts:

> ENTER # OF POINTS TO EACH SIDE (1-20) OR 0 TO EXIT.

- After the user enters the appropriate coefficient, VPG displays the averaged curve.

note: The selection of 5 averages the current point with the 5 points before and after the current point (11 points together).

FIR FILTER

- VPG prompts the user to enter a series of parameters:

> SELECT CURVES TO BE FILTERED

- After the user picks a displayed curve from the menu or by mouse, VPG prompts:

> ENTER PASSBAND FREQUENCY (0.0+ TO xxxx Hz) <DEFAULT = xxxx>

- The maximum passband frequency is determined through the equation:

  $Passband_{user}$ = (Total Number of Points on Graph / Time Duration)

  The default value is 10% of the number, **xxxx**, entered above.

> ENTER STOPBAND FREQUENCY XXXX (THE NUMBER JUST ENTERED ABOVE) TO XXXX HZ: <DEFAULT = xxxx>

- Stopband frequency is formulated:

  Stopband = $Passband_{user}$ + ($Passband_{max}$ - $Passband_{user}$) The default value depends on the $Passband_{user}$.

> ENTER MAXIMUM PASSBAND RIPPLE (0.0+ TO 1.0-dB): <DEFAULT = 0.1>

- Passband ripple value is between 0.0 and 1.0 dB.

> ENTER MINIMUM STOPBAND ATTENUATION (1.0+ dB): < DEFAULT = 10.0>

- Stopband attenuation must be greater than 1.0 dB.
- The stopband frequency and maximum passband ripple should not be too close to either the given low- or high-end limits.
- Attenuation factors commonly range between 10 and 50.
- It is recommended that the user choose a passband frequency that is approximately 5 to 20% of the number, **xxxx**, entered above. Too small a number will cause a computational error and too large a number will cause the program to stall.
- The stopband frequency and maximum passband ripple should not

Continued on next page.

be too close to either the given low- or high-end limits.

- Attenuation factors commonly range between 10 and 50.

BUTTERWORTH FILTER

- VPG prompts the user to enter a series of parameters:

> SELECT CURVES TO BE FILTERED

- The user may pick any displayed curve by mouse, then VPG prompts:

> ENTER PASSBAND FREQUENCY (0.0+ TO xxxx Hz) <DEFAULT = xxxx>

- The maximum passband frequency is determined through the equation:

  $Passband_{user}$ = (Total Number of Points on Graph / Time Duration) The default value is 10% of the passband frequency.

> ENTER STOPBAND FREQUENCY XXXX (THE NUMBER JUST ENTERED ABOVE) TO XXXX HZ: <DEFAULT = xxxx>

- Stopband frequency is formulated:

  $Stopband = Passband_{user} + (Passband_{max} - Passband_{user})$

  The default value depends on the $Passband_{user}$.

> ENTER MAXIMUM PASSBAND RIPPLE (0.0+ TO 1.0-dB): <DEFAULT = 0.1>

- Passband ripple value is between 0.0 and 1.0 dB.

> ENTER MINIMUM STOPBAND ATTENUATION (1.0+ dB): < DEFAULT = 10.0>

- Stopband attenuation must be greater than 1.0 dB.
- It is recommended that the user choose a passband frequency that is approximately 5 to 20% of the number, **xxxx**, entered above. Too small a number will cause a computational error and too large a number will cause the program to stall.
- The stopband frequency and maximum passband ripple should not be too close to either the given low- or high-end limits.
- Attenuation factors commonly range between 10 and 50.

SAE FILTER

- VPG prompts the user to enter a series of parameters:

> SELECT CURVES TO BE FILTERED

- The user may pick any displayed curve via the menu or mouse, then VPG prompts:

> ENTER CUTOFF FREQUENCY (0.0+ TO xxxx Hz) <DE-

Continued on next page.

FAULT = xxxx>

- The maximum passband frequency is determined through the equation:

  $Passband_{user}$ = (Total Number of Points on Graph / Time Duration) The default value is 10% of the number, **xxxx**, entered above.

> ENTER STOPBAND FREQUENCY XXXX (THE NUMBER JUST ENTERED ABOVE) TO XXXX HZ: <DEFAULT = xxxx>

- Stopband frequency is formulated:

  $Stopband = Passband_{user} + (Passband_{max} - Passband_{user})$

The default value depends on the $Passband_{user}$.

> ENTER MAXIMUM PASSBAND RIPPLE (0.0+ TO 1.0- dB): <DEFAULT = 0.1>

- Passband ripple value is between 0.0 and 1.0 dB.

> ENTER MINIMUM STOPBAND ATTENUATION (1.0+ dB): < DEFAULT = 10.0>

- Stopband attenuation must be greater than 1.0 dB.
- It is recommended that the user choose a passband frequency of about 5 to 20% of the number, **xxxx**, entered above. Too small a number will cause a computational error and too large a number will cause the program to stall.
- The stopband frequency and maximum passband ripple should not be too close to either the given low- or high-end limits.
- Attenuation factors commonly range between 10 to 50.

CROSS PLOT

- This command plots curves against curves. For example, if f(t), a(t), and d(t) are time history components for force, acceleration, and displacement, then, f(t) and a(t) against d(t) will generate two curves that relate force and acceleration to displacement.

RESULTANT

- This command derives a curve from selected curves according to the following calculations:

$$y(t) = \sqrt{[x_1(t)]^2 + [x_2(t)]^2 + [x_3(t)]^2 + \ldots}$$

EXTERNAL

- The user may execute the external command to manipulate the XY curves.

Continued on next page.

- The user picks a curve(s) from the display and enters a user command. (This command may be generated from a C or FORTRAN program, a shell script, or a UNIX program.) VPG gives two arguments (input and output file name) to the user command. The user command is designed to read the input file, manipulate it, and insert its result into an output file. The file names are:
  /tmp/vpg_ext_command_input_curves;
  /tmp/vpg_ext_command_output_curves.
  Both files use the list-directed format for VPG.

DONE

- The user may select DONE to execute the command.

CLEAR SCREEN

This function removes objects from the display area.

1. VPG prompts:
   ```
   ?   CLEAR CURVES
       CLEAR MODEL
       CLEAR ETA LABEL
       CLEAR PART LABEL
       CLEAR ALL
       EXIT
   ```

2. CLEAR SCREEN is handy if the user wants to remove items from the screen for hardcopy. This command may also be used to remove a model from a graph.

DELETE CURVES

This function removes curves from the active graph.

1. VPG prompts:
   ```
   >   SELECT A CURVE (LEFT: SELECT, MIDDLE: DESELECT,
       RIGHT: DONE)
   ?   [list of curves]
       UNDO
       ABORT
       DONE
   ```

2. VPG deletes the selected curves.

HARDCOPY

This function sends a postscript image of the display area to the local printer. The postscript driver must be initialized with the VPG variables before printing. The variables are contained in the **.VPGHardcopyDefault** file and installed in the **$VPG_PATH** directory. To obtain a copy, contact VPG technical support at ETA.

1. VPG prompts:
   <list of all curves>
   TOP BOTTOM
   - Allows the user to print two selected graphs in a vertical orientation.

   LEFT RIGHT
   - Allows the user to print two selected graphs in a horizontal orientation.

   4 GRAPHS
   - Prints four selected graphs.

   EXIT

OPTIONS

The OPTIONS commands allow the user to manipulate the following display settings for graphs: axis, curve, grid, label, and legend.

-OPTIONS
AXIS OPTIONS

1. VPG Prompts:
   > ENTER COMMAND
   ? AXIS COLOR
   - This command changes the text color of an axis. VPG prompts the user to select a new color from the color bar.

   AXIS TEXT
   - This command changes the text label of an axis. VPG prompts the user for a new label (a carriage return maintains the current label).

   > ENTER TEXT FOR THE X LABEL
   > ENTER TEXT FOR THE Y LABEL
   DONE
   - This command returns the user to the parent menu.

-OPTIONS
CURVE OPTIONS

CURVE OPTIONS

- VPG displays the CURVE OPTION menu and prompts:

```
>   ENTER COMMAND-
?   CURVE COLOR
```

  - This changes the color of both the curve and its name. VPG displays the color bar from which the user selects a new color.

CURVE NAME

  - VPG displays the old name and prompts the user to enter a new name:

```
>     OLD NAME: [old name]
>     ENTER NEW NAME
```

  - VPG displays the new name after the user exits the command.

CURVE MARKER

  - Displays user defined curve markers evenly across the length of the curve.

CURVE PATTERN

  - This changes the line style for the selected curve.

```
>   SELECT TYPE FROM THE MENU
?   1 - SOLID LINE
    2 - DASHED LINE
    3 - DOTTED LINE
    4 - DOTTED DASHED LINE
    5 - ABORT
```

  note: DOTTED DASHED is not available on all platforms. When this pattern is unavailable, VPG uses the solid pattern as a default.

CURVE SIZE

  - Changes the thickness of the displayed line (or dot pattern) on the graph.
  - This command is not available on all platforms (specifically HP).

DONE

  - This returns the user to the parent menu.

-OPTIONS
GRAPH

```
GRAPH
      >   SELECT A GRAPH
      ?   [list of graph names]
          CG - CURRENT GRAPH
      >   ENTER COMMAND-
      ?   CHANGE BORDER COLOR
```

- This changes the border color of the graph with a new color that the user selects from the color bar.

BORDER ON/OFF

- This toggles the border display of a graph on and off. The default setting for this function is ON.

NAME

- VPG displays the original name and prompts:

```
>   ENTER NEW NAME
```

NAME COLOR

- This changes the color of the graph name with a new color that the user selects from the color bar.

NAME LOCATION

- Defines the graph name location.

NAME SIZE

- Allows the user to set the graph name size.

DONE

- This returns the user to the parent menu.

-OPTIONS
GRID OPTIONS

```
GRID OPTIONS
      >   SELECT A GRAPH
      ?   [list of graph names]
          CURRENT GRAPH
      >   ENTER COMMAND-
      ?   GRID COLOR
```

Continued on next page.

- This replaces the color of the grid with a new color that the user chooses from the color bar. If the user changes the grid color while the grid is off, the axes and the labels change color too.

ON/OFF

- This toggles the grid display of the graph on and off. The default setting for this command is OFF.

PATTERN

- This changes the line style for the selected grid.

```
>   SELECT PATTERN
?     1 - SOLID LINE
      2 - DASHED LINE
      3 - DOTTED LINE
      4 - DOTTED DASHED LINE
      5 - ABORT
```

note: DOTTED DASHED is not available on all platforms.

SIZE

- This changes the grid spacing on the active graph.

```
>   ENTER SIZE ( 0.100 - 2.000) OR 0 TO
    EXIT
```

SCALE

```
>   ENTER COMMAND-
?   N - NONE USES LOG SCALE
    X - X USES LOG SCALE
    Y - Y USES LOG SCALE
    B - BOTH USE LOG SCALE
```

X FORMAT

- Sets grid scale to user specified x format.

Y FORMAT

- Sets grid scale to user specified y format.

DONE

- This returns the user to the parent menu.

Continues on following page.

-OPTIONS
LABEL OPTIONS

LABEL OPTIONS
> ENTER COMMAND-
? ADD LABEL

- This adds a label to the active graph.

> ENTER CHARACTER SIZE (0.0 - 10.0) <1.0>
> ENTER TEXT FOR THE LABEL
> PICK LOCATION FOR THE LABEL

note: The selection point is the center of the label.

COLOR

- This changes the color of a label with a new color that the user chooses from the color bar.

LOCATION

> PICK LABEL

- The user selects a label for relocation.

SIZE

- The user enters the new size of the label.

> ENTER SIZE (0.00 - 10.000) OR 0 TO EXIT

TEXT

> PICK LABEL

- The user selects a label for renaming and enters the new text.

> ENTER TEXT FOR THE LABEL

DELETE

> PICK LABEL

- The user selects a label to delete, then VPG prompts:

? UN - UNDO
AB - ABORT
DO - DONE

DONE

- This returns the user to the parent menu.

-OPTIONS
LEGEND

LEGEND

> SELECT A GRAPH
? [list of graph names]
CURRENT GRAPH

- The user may select a graph via the menu or mouse (while in multi-graph mode).

> ENTER COMMAND-
? ON/OFF

- This toggles the legend of the graph on and off. The default for this function is ON.

LOCATION

- VPG displays the **CHANGE LEGEND LOCATION** menu:

? UPPER LEFT
UPPER RIGHT
LOWER LEFT
LOWER RIGHT
DONE

- The user selects the new location from the menu options. To exit this function, the user selects DONE.

PATTERN ON/OFF

- This toggles the line pattern on and off in the legend of the graph. The default for this command is ON.

BORDER ON/OFF

- This toggles the border of the legend on and off in the graph. The default for this command is ON.

DONE

- This returns the user to the parent menu.

PLOT CURRENT GRAPH

This function scales the current graph to fill the display area.

PLOT MULTI GRAPHS

This function displays multiple graphs on screen.

1. VPG prompts:

```
>   ENTER COMMAND-
?   2 TOP BOTTOM
```

- The graphs are plotted as two horizontal tiles.

```
>   SELECT GRAPHS
?   [list of available graphs]
    UNDO
    ABORT
    DONE
```

```
2 LEFT RIGHT
```

- The graphs are plotted as two vertical tiles.

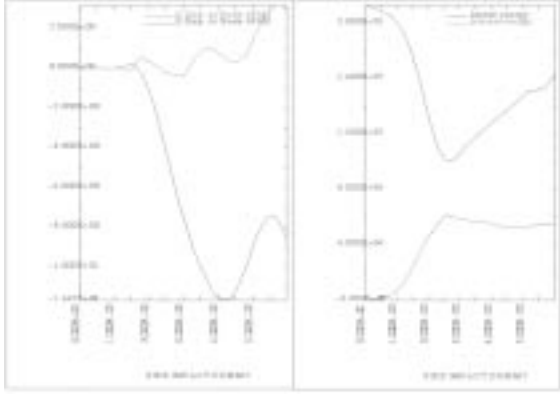
```
>   SELECT GRAPHS
?   [list of available graphs]
    UNDO
    ABORT
    DONE
```

```
4 GRAPHS
```

- The graphs are plotted as four square tiles (2 by 2).

```
>   Select graphs
?   [list of available graphs]
    UNDO
    ABORT
    DONE
```

Two graphs are displayed on screen in the 2 LEFT RIGHT format.



Continued on next page.

16 GRAPHS

- The graphs are plotted in a 16-square tile display (4 by 4).

```
>   SELECT GRAPHS
?   [list of available graphs]
    UNDO
    ABORT
    DONE
```

ARBITRARY

- The user determines the graph size by positioning the cursor on screen and dragging it downward and to the right with the mouse until the desired area is reached.

```
>   SELECT GRAPHS
```

- VPG displays all files in memory.

```
?   [list of available graphs]
    UNDO
    ABORT
    DONE
>   DEFINE WINDOW
```

- The user determines the size and location of the window via the mouse.

FIXED SIZE

- This command is similar to the arbitrary format used to define a window except that the same size format duplicates the first window for each subsequent graph. Users simply position the cursor and press the mouse button in the upper left corner of the area where they wish to display a new graph.

```
>   SELECT GRAPHS
?   [list of available graphs]
    UNDO
    ABORT
    DONE
>   DEFINE WINDOW
```

- The user determines the size and location of the window via the mouse.

```
>   SELECT GRAPHS
```

- The user displays the next graph by locating the upper left corner of its window via the mouse.

DONE

QUERY

This command displays user information about a graph, a curve or a point.

1. VPG Prompts:

? GRAPH

> SELECT A GRAPH

? [list of graph names]
CURRENT GRAPH

- The user may choose from the menu or pick from the graphics area if in multi-graph mode.
- VPG displays the name of the graph and the number of curves in the graph.

> name: [graphname] # of curves: [xxxx]

CURVE

> SELECT A GRAPH

? [list of graph names]
CURRENT GRAPH

- The user may choose from the menu or pick from the graphics area if in multi-graph mode.
- VPG displays the name of the curve, the number of points on the curve, the X range, and the Y range.

> NAME:[filename] # of Pt: [xxxx] [xxxx]<= x <= [xxxx] [xxxx]<= y <= [xxxx]

- The user chooses another curve and VPG changes the statistics in the display menu.

POINT

- The user selects a point on a curve and VPG returns the X, Y coordinates of the point.

> x: [xxxx] y: [xxxx]

? DONE

DONE

RANGE

This function zooms in on a specific section of the graph.

1. VPG prompts:
   > ENTER XMIN, XMAX
   > ENTER YMIN, YMAX
   - After the user enters the Y values, VPG updates the display to reflect the zoom.
   - **RANGE** is a toggle switch. The first toggle initiates the zoom, and the second turns off the zoom and returns the display to the original view.

READ DATABASE

This command downloads a variety of database files.

note: The **READ DATABASE** command requires several levels of submenus.

1. VPG prompts the user to select a display method for the XY Plot:
   > SELECT GRAPH TYPE TO PLOT
   ? NEW GRAPH
   - This creates new graph.

   CURRENT GRAPH
   - This utilizes a current working graph (if it exists).

   SELECT GRAPH
   - This selects an existing graph to hold the new curves.

   NO GRAPH
   - This keeps the curve information in memory without displaying it.

2. Depending on which option the user selects, VPG displays a series of menu options. These options are described in the following pages.

Continued on next page.

-READ DATABASE
NEW GRAPH

The functions in the NEW GRAPH submenu create a new graph.

1. VPG prompts:

```
>   ENTER DATABASE TYPE:
?   NODAL POINT DATA
    GLOBAL DATA
    MATERIAL ENERGIES
    RESULTANT INTERFACE FORCE
    DISCRETE ELEMENT
    RIGID WALL FORCE
    CROSS SECTION FORCE
    AIRBAG STATISTICS
    INTERFACE ENERGIES
    RIGID BODY DATA
    JOINT FORCE
    BRICK ELEMENT DATA
    BEAM ELEMENT DATA
    SHELL ELEMENT STRAIN
    SHELL ELEMENT STRESS
    NODAL FORCE
    TAURUS DATABASE
    LIST DIRECTED
    COLUMN DIRECTED
    EXIT
```

2. Depending on which option the user selects, VPG displays a series of menu options. These options are described in the following pages.

-READ DATABASE
- NEW GRAPH
NODAL POINT DATA

This function plots nodal point data from the **<nodout>** file.

1. VPG displays the VPG FILE MENU window and prompts for a **<nodout>** file.

2. VPG displays the SELECT NODES menu and prompts:
   > SELECT A NODE (LEFT: SELECT, MIDDLE: DESELECT, RIGHT: DONE)
   ? <list of all nodes>
   UNDO
   ABORT
   DONE
   - The user has the option of selecting nodes on the model by mouse pick or from the list of nodes in the VPG MENU window.,
   - The user can select multiple nodes on the model. If the node is not in the database, VPG displays:
     > INVALID NODE
     and waits for the user to enter a different selection.

3. After selecting done, VPG displays the NODAL GRAPH menu.
   ? X DISP. VS. TIME
   Y DISP. VS. TIME
   Z DISP. VS. TIME
   X VELO VS. TIME
   Y VELO VS. TIME
   Z VELO VS. TIME
   X ACC VS. TIME
   Y ACC VS. TIME
   Z ACC VS. TIME
   UNDO
   ABORT
   DONE
   - Once the type of graph is selected, it is displayed in the menu window.

-READ DATABASE
-NEW GRAPH
GLOBAL DATA

This function plots the global time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME

2. Once the file has been entered, VPG displays the GLOBAL TIME HISTORY menu and prompts:

```
>   SELECT VARIABLES
?   TIME STEP
    KINETIC ENERGY
    INTERNAL ENERGY
    SPRING & DAMPER ENERGY
    HOURGLASS ENERGY
    SYSTEM DAMPING ENERGY
    SLIDING INTERFACE ENERGY
    EXTERNAL WORK
    TOTAL ENERGY
    TOTAL ENERGY/INITIAL ENERGY
    GLOBAL X VELOCITY
    GLOBAL Y VELOCITY
    GLOBAL Z VELOCITY
    UNDO
    ABORT
    DONE
```

   - This menu is dependent on the glstat file. In most cases, the options are a subset of the above list.
   - The user may select multiple plot types in this menu.
   - After the user selects DONE, VPG displays the curves and returns the user to the TIME HISTORY PLOT menu.

Continued on next page.

-READ DATABASE
-NEW GRAPH
MATERIAL ENERGIES

This function plots material time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME
   • VPG displays the **SELECT MAT** menu and prompts:
   > SELECT MAT:
   ? [displays available materials]
   UNDO
   ABORT
   DONE

2. VPG displays the MATERIAL TIME HISTORY menu:
   ? INTERNAL ENERGY
   KINETIC ENERGY
   X MOMENT
   Y MOMENT
   Z MOMENT
   X VEL. RIGID BODY
   Y VEL. RIGID BODY
   Z VEL. RIGID BODY
   UNDO
   ABORT
   DONE

3. After the user selects DONE, VPG displays the curves and returns to the GRAPH menu.

-READ DATABASE
-NEW GRAPH
RESULTANT INTERFACE FORCE

This function plots resultant interface time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME
   • VPG displays the SELECT INTERFACE menu and prompts:
   > SELECT interface:
   ? [lists available slave and master numbers]
   UNDO
   ABORT
   DONE

2. After the user selects DONE, VPG displays the RESULTANT INTERFACE FORCE menu and prompts:
   > SELECT TYPE OF TIME HISTORY PLOT
   ? X
   Y
   Z
   UNDO
   ABORT
   DONE

3. After the user selects DONE, VPG displays the curves and returns to the TIME HISTORY menu.

-READ DATABASE
-NEW GRAPH
DISCRETE ELEMENT

This function plots discrete element time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME
   • VPG displays the **SELECT SPRING/DAMPER** menu and prompts:
   > SELECT Spring/Damper:

Continued on next page.

```
?   [lists available spring/damper Element numbers]
    UNDO
    ABORT
    DONE
```

2. After the user selects DONE, VPG displays the DISC. ELEM. TIME HISTORY menu:

```
?   X FORCE
    Y FORCE
    Z FORCE
    RESULTANT FORCE
    UNDO
    ABORT
    DONE
```

3. After the user selects DONE, VPG displays the curves and returns to the GRAPH menu.

-READ DATABASE
-NEW GRAPH
RIGID WALL FORCE

This function plots rigid wall time history data.

1. VPG displays the VPG file menu and prompts:

```
>   ENTER VPG GRAPH FILE NAME
```

   - VPG displays the **SELECT WALLS** menu and prompts:

```
>   SELECT WALLS
?   UNDO
    ABORT
    DONE
```

   - VPG displays the wall numbers that are in the database.

2. After the user selects DONE, VPG displays the RIGID WALL TIME HISTORY menu, and prompts:

```
>   SELECT TYPE OF TIME HISTORY PLOT
?   NORMAL FORCE
    X FORCE
```

Continued on next page.

```
Y FORCE
Z FORCE
UNDO
ABORT
DONE
```

3. After the user selects DONE, VPG displays the curves and returns to the GRAPH menu.

-READ DATABASE
-NEW GRAPH
CROSS SECTION FORCE

This function plots the cross section time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME
   - VPG displays the SELECT SECTIONS menu and prompts:

   > SELECT SECTIONS:
   ? [displays available section numbers]
   UNDO
   ABORT
   DONE

2. After the user selects DONE, VPG displays the CROSS SEC. TIME HISTORY menu and prompts:
   > SELECT TYPE OF TIME HISTORY PLOT
   ? X FORCE
   Y FORCE
   Z FORCE
   TOTAL FORCE
   X MOMENT
   Y MOMENT
   Z MOMENT
   TOTAL MOMENT
   X CENTROID
   Y CENTROID
   Z CENTROID
   AREA

Continued on next page.

UNDO
ABORT
DONE

3. After the user selects DONE, VPG displays the curves and returns to the GRAPH menu.

-READ DATABASE
-NEW GRAPH
AIRBAG STATISTICS

This function plots the airbag time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME
   • VPG displays the **SELECT AIRBAG** menu and prompts:
   > SELECT AIRBAGS:
   ? [displays available Airbag numbers and]
   UNDO
   ABORT
   DONE

2. After the user selects DONE, VPG displays the AIR BAG TIME HISTORY menu:
   ? AIR BAG TIME HIS.
   VOLUME
   PRESSURE
   INTERNAL ENERGY
   DM/DT IN (change in mass/change in time)
   DENSITY
   DM/DT OUT (change in mass/change in time)
   TOTAL MASS
   GAS TEMPERATURE
   UNDO
   ABORT
   DONE

3. After the user selects DONE, VPG displays the curves and returns to the GRAPH menu.

-READ DATABASE
-NEW GRAPH
INTERFACE ENERGIES

This function plots the interface time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME
   - VPG displays the **SELECT INTERFACE** menu and prompts:

   > SELECT INTERFACE ENERGIES:
   ? <list of all slaves>
   <list of all masters>
   TOTAL SLAVE SIDE
   TOTAL MASTER SIDE
   TOTAL ENERGY
   UNDO
   ABORT
   DONE
   - VPG displays the Slave and Master energies available in the database.

2. After the user selects DONE, VPG displays the curves and returns to the GRAPH menu.

-READ DATABASE
-NEW GRAPH
RIGID BODY DATA

This function plots the rigid body time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME
   - VPG displays the SELECT RIGID BODY MENU and prompts:

   > SELECT Rigid body:
   ? [displays available rigid body numbers]
   UNDO
   ABORT
   DONE

2. After the user selects DONE, VPG displays the RIGID BODY TIME HISTORY menu:

Continued on next page.

```
?   RIGID BODY TIME HIS.
    X DISPLACEMENT
    Y DISPLACEMENT
    Z DISPLACEMENT
    X ROTATION
    Y ROTATION
    Z ROTATION
    X VELOCITY
    Y VELOCITY
    Z VELOCITY
    XRVELOCITY
    YRVELOCITY
    ZRVELOCITY
    X ACCELERATION
    Y ACCELERATION
    Z ACCELERATION
    XRACCELERATION
    YRACCELERATION
    ZRACCELERATION
    UNDO
    ABORT
    DONE
```

3. After the user selects DONE, VPG displays the curves and returns to the GRAPH menu.

-READ DATABASE
-NEW GRAPH
JOINT FORCE

This function plots the joint force time history data.

1. VPG displays the VPG file menu and prompts:
```
>   ENTER VPG GRAPH FILE NAME
```

2. Once the file has been entered, VPG displays the SELECT JOINT FORCE menu in the VPG menu window.
```
>   SELECT FORCE DATA
    <list of forces>
    UNDO
    ABORT
    DONE
```

3. After selecting the forces, VPG prompts:

```
>   SELECT TYPE OF TIME HISTORY PLOT
    X FORCE
    Y FORCE
    Z FORCE
    X MOMENT
    Y MOMENT
    Z MOMENT
    RESULTANT FORCE
    RESULTANT MOVEMENT
    UNDO
    ABORT DONE
```

   - Once the plot type is selected, VPG plots the graph.

-READ DATABASE
-NEW GRAPH

BRICK ELEMENT DATA

This function plots the brick element time history data.

1. VPG displays the VPG file menu and prompts:

```
>   ENTER VPG GRAPH FILE NAME
```

2. Once the file has been entered, VPG displays the SELECT ELEMENT DATA menu in the VPG menu window.

```
    <list of elements>
    UNDO
    ABORT
    DONE
```

3. After selecting the elements, VPG prompts:

```
>   SELECT TYPE OF TIME HISTORY PLOT
    AXIAL
    SHEAR
    SHEAR T
    MOMENT S
    MOMENT T
    TORSION
    UNDO
    ABORT DONE
```

   - Once the plot type is selected, VPG plots the graph.

-READ DATABASE
-NEW GRAPH

BEAM ELEMENT DATA

This function plots the beam element time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME

2. Once the file has been entered, VPG displays the SELECT ELEMENT DATA menu in the VPG menu window.
   > SELECT ELEMENT DATA
   <list of elements>
   UNDO
   ABORT
   DONE

3. After selecting the elements, VPG prompts:
   > SELECT TYPE OF TIME HISTORY PLOT
   AXIAL
   SHEAR
   SHEAR T
   MOMENT S
   MOMENT T
   TORSION
   UNDO
   ABORT DONE
   - Once the plot type is selected, VPG plots the graph.

-READ DATABASE
-NEW GRAPH

SHELL ELEMENT STRAIN

This function plots the shell element strain time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME

2. Once the file has been entered, VPG displays the SELECT ELEMENT DATA menu in the VPG menu window.
   > SELECT ELEMENT DATA

Continued on next page.

```
<list of elements>
UNDO
ABORT
DONE
```

3. After selecting the elements, VPG prompts:

```
>   SELECT TYPE OF TIME HISTORY PLOT
    AXIAL
    SHEAR
    SHEAR T
    MOMENT S
    MOMENT T
    TORSION
    UNDO
    ABORT DONE
```

   - Once the plot type is selected, VPG plots the graph.

-READ DATABASE
-NEW GRAPH

SHELL ELEMENT STRESS

This function plots the shell element stress time history data.

1. VPG displays the VPG file menu and prompts:

```
>   ENTER VPG GRAPH FILE NAME
```

2. Once the file has been entered, VPG displays the SELECT ELEMENT DATA menu in the VPG menu window.

```
>   SELECT ELEMENT DATA
    <list of elements>
    UNDO
    ABORT
    DONE
```

3. After selecting the elements, VPG prompts:

```
>   SELECT TYPE OF TIME HISTORY PLOT
    AXIAL
    SHEAR
    SHEAR T
    MOMENT S
```

Continued on next page.

MOMENT T
TORSION
UNDO
ABORT DONE

- Once the plot type is selected, VPG plots the graph.

-READ DATABASE
-NEW GRAPH
NODAL FORCE

This function plots the nodal force time history data.

1. VPG displays the VPG file menu and prompts:
   > ENTER VPG GRAPH FILE NAME

2. Once the file has been entered, VPG displays the SELECT FORCE menu in the VPG menu window.
   > SELECT FORCE DATA
   <list of forces>
   UNDO
   ABORT
   DONE

3. After selecting the elements, VPG prompts:
   > SELECT TYPE OF TIME HISTORY PLOT
   AXIAL
   SHEAR
   SHEAR T
   MOMENT S
   MOMENT T
   TORSION
   UNDO
   ABORT DONE
   - Once the plot type is selected, VPG plots the graph.

-READ DATABASE
-NEW GRAPH
TAURUS DATABASE

Currently not supported.

-READ DATABASE
-NEW GRAPH
LIST DIRECTED

This function creates a graph in a list directed (neutral) format according to user specifications.

note: The graphs in the GRAPH menu are automatically saved in the list directed format.

1. VPG displays a list of files ending with the extension (.lst). The user may select files from the menu by mouse or keyboard. UNIX commands are also available as outlined in the manual.

2. After the user selects a file, VPG returns to the TIME HISTORY PLOT menu.

List Directed Format

```
RECORD 1          :N
RECORD 2          :N1 'TITLE'
RECORD 3          :XDATA1     YDATA1
                .
                .
                .
RECORD 2 + N1     :NDATAN1     YDATAN1
RECORD 3 + N1     :N2 'TITLE FOR CURVE2'
RECORD 4 + N1     :XDATA1 YDATA1
                .
                .
                .
etc.
```

Where:

```
N            = NUMBER OF CURVES
N1           = NUMBER OF POINTS ON THE FIRST CURVE
N2           = NUMBER OF POINTS ON THE SECOND CURVE
TITLE        = TITLE WITH A MAXIMUM OF 20 CHARACTERS
```

-READ DATABASE
-NEW GRAPH
COLUMN DIRECTED

This function creates a graph in column directed (neutral) format according to user specifications. Column directed format allows the user to plot multiple sets of Y data for the same X data (i.e. material energy vs. time for three different parts).

1. VPG displays a list of all files ending with the extension (.col) in the options area. The user may select files from the menu by mouse or keyboard.

2. After the user selects a file, VPG returns to the TIME HISTORY PLOT menu.

Column Directed Format

```
REC1          :N
REC2          :NP
REC3          :'TITLE 1'
.
.
.
REC2+N        :'TITLE N'
REC3+N        :X Y1 Y2 Y3 ...YN
.
.
.
REC2+N+NP     :XNP Y1NP Y2NP Y3NP ...YNNP
```

Where:

```
N           = NUMBER OF CURVES OR Y COLUMNS
NP          = NUMBER OF POINTS IN CURVES
TITLE       = TITLE WITH A MAXIMUM OF 20 CHARACTERS.
```

-READ DATABASE
CURRENT GRAPH

This function creates an additional curve(s) in the current graph.

Continued on next page.

1. CURRENT GRAPH
   ```
   >   ENTER DATABASE TYPE
   ?   ASCII DYNA3D DATABASE
       EXIT
   ```

2. These commands are described under the heading, TIME HISTORY PLOTS, in the ASCII DYNA3D DATABASE and VPG CURVE FILES sections of the *User's Manual*.

-READ DATABASE
SELECT GRAPH

This function creates a new graph and places it in a selected graph.

1. VPG lists available graphs in the menu area. Once the desired graph has been selected, VPG prompts:
   ```
   >   ENTER DATABASE TYPE
   ?   ASCII DYNA3D DATABASE
       EXIT
   ```

2. These commands are described under the heading, TIME HISTORY PLOTS, in the ASCII DYNA3D DATABASE and VPG CURVE FILES sections of the *User's Manual*.

-READ DATABASE
NO GRAPH

This function saves the database in memory without displaying the curves.

1. **NO GRAPH**
   ```
   >   ENTER DATABASE TYPE
   ?   ASCII DYNA3D DATABASE
       EXIT
   ```

2. These commands are described under the heading, TIME HISTORY PLOTS, in the ASCII DYNA3D DATABASE and VPG CURVE FILES sections of the *User's Manual*.

SAVE CURVES

This function saves changes in curves that the user may recall at a later time. The curves are saved in list directed format. (List and Column directed format are covered in the XY Plot section of the manual.)

1 VPG Prompts:

```
> SELECT AND SAVE CURVES
    ? UNDO
      ABORT
      DONE
    > ENTER FILE NAME:
```

- If the filename already exists, VPG asks if the user wants to overwrite it.

```
  CURVES IN ONE GRAPH
    > LIST OF GRAPHS
    ? CURRENT GRAPH
      EXIT
    > ENTER FILE NAME
```

- If the filename already exists, VPG asks if the user wants to overwrite it.

```
  CURVES IN CURRENT GRAPH
    > ENTER FILE NAME:
```

- If the filename already exists, VPG asks if the user wants to overwrite it.

```
  ALL CURVES
    > ENTER FILE NAME:
```

- If the filename already exists, VPG asks if the user wants to overwrite it.

```
  DONE
```

SET DEFAULTS

This function sets default values for graphs and curves. The default values take effect for newly-loaded databases.

SELECT GRAPH

This function selects a current graph from the database. VPG displays a menu of the graphs that are in the memory. The user selects graphs from this menu as necessary.

TIME <-->FREQUENCY

The TIME <-->FREQUENCY function allows the user to convert time domain to frequency domain response for signal processing analysis. The equations used to convert time domain to frequency domain and frequency domain to time domain are represented here where w=frequency and t=time:

$$H(\omega) = \int_{-\infty}^{\infty} h(t) e^{i\omega t} dt$$

$$h(t) = \frac{1}{2\pi} \int_{-\infty}^{\infty} H(\omega) e^{-i\omega t} d\omega$$

A TIME <-->FREQUENCY (**.fre**) file is created by converting a curve file to the **.fre** format using the FAST FOURIER TRANSFORM equation. The **.fre** file is a two or three column format file with the first column representing frequency, the second column representing amplitude or power spectrum density and the third optional column, phase angle data.

1. If the user has not yet read a graph into the database, GRAPH is selected from the MAIN MENU. (If the user already has a graph displayed, skip to step 5).

2. The VPG FILE MENU is displayed and VPG prompts for a file name.

3. Select CREATE NEW FILE from the VPG FILE MENU. VPG displays a window for the curve.

4. Select READ DATABASE from the GRAPH menu. Once a database has been read and displayed in the graph window the TIME<-->FREQUENCY function can be utilized. The TIME<-->FREQUENCY function can be used with any type of graph data.

5. After selecting the TIME<-->FREQUENCY function to translate time to frequency, VPG prompts:

   > CHANGE TO FREQUENCY DOMAIN

   > DO YOU WANT DATA WINDOWING FUNCTION (Y/N)?

   - YES causes the curves to be smoother.

   > DO YOU WANT TO OUTPUT PSD OR AMPLITUDE (P/A)?

   - This defines the second column in the output file. PSD is the power spectrum density.

UTILITY

UTILITY includes the following sections: VIEWING OPTIONS, VIEWING OPTIONS WINDOW, PART CONTROL, SETUP, UTILITY, and DISPLAY PARAMETER OPTIONS WINDOW. The functions in these sections allow the user to alter the model's display, define the model's parameters, and define the model's part makeup.

VIEWING OPTIONS

The functions in the VIEWING OPTIONS menu are used to adjust the display of the items on the screen.

VIEWING OPTIONS MENU

CHANGE COLORS
FILL COLOR ON/OFF
FILL HIDE PLOT
MAGNIFY WINDOW
REPEAT
REVERSE ROTATION
RECALL VIEW
SAVE VIEW
VIEWPORTS ON/OFF
SCREEN AXES ROT.
VIRTUAL AXES ROT.
DEFINE TITLE
CURSOR ZOOM
TRUE VIEW
DEPTH CUEING
FLIP NORMALS
HIDDEN SURFACE REM.
TRANSP. MODE ON/OFF
MOVE LIGHT SOURCE
ACTIVE WINDOW
EXIT

A detailed description of each function is given in the following sections.

CHANGE COLORS

This function restores the color bar to its original default values. The RESET COLOR MAP may be viewed through the PLOT COLOR BAR command.

1. **CHANGE COLORS**
   - The functions in this submenu allow the user to change the color of any displayed item (including part colors). A full description of each option is given below.

```
CHANGE PART COLORS
CHANGE COLOR MAP
RESET COLOR MAP
PART COLORS ON/OFF
FILL COLOR ON/OFF
COLOR BAR
EXIT
```

-CHANGE COLORS
CHANGE PART COLORS

This function changes the colors of selected parts. The user may activate a color change when the parts are on or off.

1. VPG prompts:
   ```
   > SELECT PART TO CHANGE COLOR
   > PICK A LINE (L) AN ELEM (C) OR PART NAME (P) OF A
     PART
   ```
   - The user selects a part as follows:
     A. by placing the cursor on a displayed part line and typing the letter (L).
     B. by placing the cursor on a displayed part element and pressing the left mouse button.
   - VPG displays the part list from which the user may select any part.

2. Once the user selects a part, VPG prompts:
   ```
   > SELECT COLOR FROM COLOR BAR OR EXIT
   ```
   - VPG displays the color bar on the right edge of the screen. The user may select a desired color via the mouse.

3. EXIT terminates this command.

-CHANGE COLORS
CHANGE COLOR MAP

This function changes the default colors of the COLOR BAR.

1. VPG prompts:
   > SELECT COLOR FROM COLOR BAR OR EXIT

2. Once the user selects a color with the left mouse button, VPG prompts:
   > ORIGINAL RGB VALUES: rrr ggg bbb
   > ENTER NEW RGB VALUES (0 - 255)
   - i.e., 255,0,0 for RED; 0,255,0 for GREEN; 0,0,255 for BLUE.
   - note: "," (commas) are required.

-CHANGE COLORS
RESET COLOR MAP

The functions in this menu allow the user to change the color of any displayed item (including part colors) in the VPG window.

-CHANGE COLORS
PART COLORS ON/OFF (toggle)

This function changes each element type into a separate and uniform color (e.g., quadrilateral elements turn blue; triangular elements turn red; 8-node cube elements turn green; 6-node wedge elements turn magenta; and 4-node tetrahedron elements turn light magenta). This function activates the color changes when selected once, and restores the original colors when selected twice.

-CHANGE COLORS
FILL COLOR ON/OFF (toggle)

The FILL COLOR ON/OFF function fills the displayed elements with a designated color when selected once, and returns the elements to their original color when selected again. This function, when used alone, is unable to accurately represent the depth perspective of a model. The parts may appear to warp or penetrate each other. However, the FILL COLOR command displays an accurate 3D perspective of the parts when used in conjunction with the HIDDEN SURFACE REMOVAL option that is described later in this section.

-CHANGE COLORS

COLOR BAR

Once the user executes this command, VPG displays the color bar and the assigned colors and numbers on the right edge of the screen. VPG removes the color bar when the user activates the REDRAW or CLEAR command.

FILL COLOR ON/OFF (toggle)

For a complete description of this function see FILL COLOR under CHANGE COLORS above.

FILL HIDE PLOT (toggle)

This function creates a hidden surface plot which displays the model in a realistic view.

note: HIDDEN SURFACE REMOVAL ON (located in the VIEWING OPTIONS menu) must be displayed at the bottom of the screen for FILL HIDE to work properly.

note: FILL HIDE PLOT is usually used in conjunction with the FILL COLOR ON/OFF command.

MAGNIFY WINDOW

This function magnifies an object in the display area.

1. VPG prompts:
   > ENTER MAGNIFICATION
   - The user may enter any positive number as the magnification factor. If the number is greater than 1, it will cause the size of the object in the display area to be enlarged. If the number is less than 1, it will cause the size of the object in the display area to be reduced, e.g., an entry of 5 will enlarge the picture, whereas an entry of .5 will reduce the picture.

REPEAT

This function is used in conjunction with the SCREEN AXES ROTATION and the VIRTUAL AXES ROTATION commands to repeat the degree of rotation and to replot the geometry of a part. The user may repeat this command as many times as desired.

REVERSE ROTATION

This function works in conjunction with SCREEN AXES ROTATION and VIRTUAL AXES ROTATION to reverse the degree of rotation and to replot the geometry of a part. The user may repeat this command as many times as desired.

RECALL VIEW

This function allows the user to recall any previously saved view. VPG contains the following predefined views.

1. **TOP VIEW**
2. **SIDE VIEW**
3. **REAR VIEW**
4. **ISOMETRIC VIEW** (60 DEGREE ISOMETRIC)

When the user selects any of the above views, VPG re-scales the displayed part accordingly.

SAVE VIEW

This function allows the user to save and store a desired view in the current VPG database.

1. VPG prompts:
   > ENTER VIEW TITLE (UP TO 18 CHARACTERS)
   - The user enters a title for the desired view. RECALL VIEW lists the new view and its corresponding view number. The program allows the user to save up to six new views plus the four views that are predefined (a total of 10 views).

Continued on next page.

2. If the user exceeds the total number of views allowed, the prompt at the bottom of the screen will read:

   > SELECT VIEW TO OVERWRITE (5 to 10)
   ? THE LIST OF YOUR SAVED VIEWS WILL BE LISTED IN THE OPTIONS AREA.

   • The user may select a view to overwrite and repeat step 1.

VIEW PORTS ON/OFF

This function allows the user to view an object using the four predefined views available in the split-screen display mode. 1 - corresponds to the default view; 2 - corresponds to the TOP VIEW (X-Y PLANE) and the SIDE VIEW (X-Z PLANE); and 4 - corresponds to the TOP VIEW (X-Y PLANE), the SIDE VIEW (X-Z PLANE), the ISOMETRIC VIEW and the REAR VIEW (Y-Z PLANE). Once the user executes the above option, the displayed model will automatically re-scale itself to fill the split-screen display.

SCREEN AXES ROTATION

This function allows the user to rotate an object in the display area according to an angle specified via the keyboard. The screen axes are fixed at all times and are defined as follows:

1. Screen X - axis is the direction from the left to right of the terminal screen.
2. Screen Y - axis is the direction from the bottom to the top of the terminal screen.
3. Screen Z - axis is the direction from the screen to the user.

The user defines the rotation by entering the angle of a particular axis. The angle is defined as positive if the rotation corresponds to a counterclockwise direction (i.e., follows the right hand rule).

1. VPG prompts:

   > ENTER 3 ANGLES TO ROTATE

   • The user may now enter the desired X, Y, Z angles of rotation.

note: The functions REPEAT ROTATION and REVERSE ROTATION are especially useful after executing the above command.

VIRTUAL AXES ROTATION

This function allows the user to rotate an object about the global (or virtual) X, Y, and Z axes according to an angle specified via the keyboard.

1. VPG prompts:
   > ENTER 3 ANGLES TO ROTATE
   - The user enters the desired global X, Y, Z angles of rotation.

note: The functions REPEAT ROTATION and REVERSE ROTATION are especially useful after executing the above command.

DEFINE TITLE

This function allows the user to enter a title or a text on the display screen.

1. VPG prompts:
   > INPUT CHARACTER SIZE (1.0 - 10.0)

2. Once the user selects the text size, VPG prompts:
   > DEFINE TITLE STARTING LOCATION
   ? ABORT
   CURSOR LOCATION
   DONE
   EXIT
   UNDO LAST
   - The user places the text by positioning the cursor at the desired location and clicking the left mouse button.

3. VPG prompts:
   > ENTER TITLE
   - The user types in the text for the title and presses enter. The text appears at the previously selected location.

CURSOR ZOOM

This function allows the user to zoom in on any displayed part.

1. VPG prompts:
   > DEFINE ZOOM WINDOW
   - The user defines the upper left corner of the zoom window by positioning the cursor on the display screen. The user then presses the left mouse button and drags the cursor diagonally down the screen until the desired window size is reached. The user clicks the mouse once to activate the window (highlighted in white).

2. VPG prompts:
   > ACCEPT ZOOM WINDOW? (YES/NO/ABORT)
   ? YES
   NO
   ABORT
   - YES rescales the part to accommodate the zoom window.
   - NO prompts the user to define a new zoom window.
   - ABORT exits the zoom option.

TRUE VIEW

This function allows the user to display an object in true view, i.e, the normal view of the local W-axis as projected onto the local UV plane.

1. VPG prompts:
   > CREATE LOCAL COORDINATES
   > PICK NODE/POINT FOR ORIGIN
   - VPG asks for a desired coordinate system (local or global).

2. VPG displays the coordinates on screen and prompts:
   > ACCEPT? (Y/N/A)
   - NO returns the user to step 2.
   - YES continues this command to step 4.
   - ABORT

3. VPG prompts:
   > KEEP ANY GLOBAL AXIS VERTICAL? (Y/N)
   ? YES

Continued on next page.

> ENTER THE VERTICAL GLOBAL AXIS (x,y,z)

- The user may select an axis to be displayed from the bottom to the top of the screen. The object remains in its true view if this option is selected.

NO

- NO displays the object in its true view (perpendicular to the W-axis).
- This option displays the desired geometry in its true view according to the global X, Y, Z coordinates or to the Z-axis (displayed vertically).

4. VPG prompts:

> DO YOU WISH TO MAP ANY LINE ON TO THE X-Y PLANE?
? YES
NO (exits the TRUE VIEW function)

5. If the user selects YES, then VPG prompts:

> SELECT LINES
? ABORT
DONE
EXIT
KEYBOARD ENTRY
LINE
PART
> SELECT TARGET PART, PICK A LINE (L) AN ELEM (C)
> OR PART NAME (P) OF A PART.

- The user selects a target part as follows:
  A. by placing the cursor on a displayed part line and typing the letter (L).
  B. by placing the cursor on a displayed element and pressing the left mouse button.

MULTI-POINT REGION
REJECT LAST
WINDOW SELECT

6. Selecting DONE copies the selected lines along the XY plane in true view .

DEPTH CUEING (toggle)

The DEPTH CUEING function simulates depth perspective, a phenomenon that causes distant objects to appear dimmer than closer objects. When the user activates the DEPTH CUEING command, VPG displays the parts of the model that are intended to appear farther from the viewer at a lower intensity.

FLIP NORMALS

The FLIP NORMAL command allows the user to reverse or "flip" the element normals during the SHADING process. Elements have a front and a back face. During the shading process, some elements may appear to be unshaded or black in color. This occurs because the element normal is in the reverse direction of the elements that reflect color. When the FLIP NORMAL command is executed, the elements that are unshaded become shaded.

note: The FLIP NORMALS command is for display purposes only. The elements that have been reversed return to their normal configuration once the user exits the FLIP NORMALS command.

HIDDEN SURFACE REMOVAL (toggle)

This function improves the integrity of the 3D simulation of a model. The user may toggle the hidden lines on and off while using the SHADING and FILL HIDE PLOT commands. This effect creates opaque elements that prevent objects in the background from showing through objects in the foreground.

1. VPG prompts:
   > HIDDEN SURFACE REMOVAL ON or
   > HIDDEN SURFACE REMOVAL OFF

note: When checking for penetration, make sure the flag HIDDEN SURFACE REMOVAL=ON appears along the bottom of the display area.

TRANSPARENCY MODE ON/OFF (toggle)

This function toggles the transparency mode on or off during shading.

1. VPG prompts:
   > TRANSPARENCY MODE ON or
   > TRANSPARENCY MODE OFF

MOVE LIGHT SOURCE

VPG uses two types of light sources: ambient light and directional light.

Ambient light comes from all directions and illuminates objects, regardless of their position or orientation, with the same intensity.

Directional light comes from a specific direction. It flows uniformly and maintains a consistent intensity. It has a directional vector that determines the face on which an object receives light.

The MOVE LIGHT SOURCE command positions a directional vector at two points that are perpendicular to an object. The right and left arrow keys move the light source in the positive and negative X-direction, while the up and down arrow keys move the light source in the positive and negative Y-direction.

ACTIVE WINDOW

This command allows the user to isolate a portion of the displayed geometry/model for more detailed viewing or editing. The user defines the region by dragging a window over the desired area or by specifying the volume of space via the keyboard with X,Y,Z coordinates or two diagonal points/nodes. VPG displays the elements, lines, and surfaces within the window (volume in space) as active. Other objects on screen remain masked and inactive.

note: An ACTIVE WINDOW may be defined at any time within any menu of VPG by clicking the right mouse button.

1. VPG prompts:

```
>   SELECT OPTION FOR ACTIVE WINDOW
?   DIAGONAL NODE/PTS
    KEY IN XYZ RANGE
    ACTIVE WINDOW OFF
    SCALE ACTIVE WINDOW
    DRAG WINDOW
    EXIT
```

-ACTIVE WINDOW
DIAGONAL NODE/POINTS

The user selects two nodes/points in a diagonal fashion to define the active widow.

-ACTIVE WINDOW
KEY IN XYZ RANGE

The user defines the size (volume) of the active window by entering its global coordinates via the keyboard. The user may also expand an existing active window by modifying any or all of its coordinates.

> ENTER XYZ RANGE (XMIN, XMAX, YMIN, YMAX, ZMIN, ZMAX)

-ACTIVE WINDOW
ACTIVE WINDOW IS OFF

- Turns off or deactivates the current active window.

-ACTIVE WINDOW
SCALE ACTIVE WINDOW

- This allows the user to scale the current active window to a larger or smaller window.

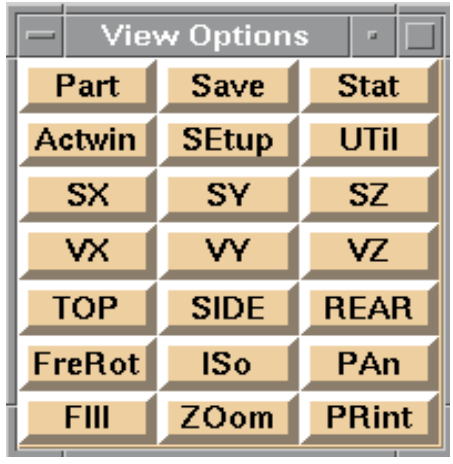
> ENTER SCALE FACTOR

-ACTIVE WINDOW
DRAG WINDOW

The user defines the active window by selecting a two-region diagonal on screen.

VIEWING OPTIONS WINDOW

The VIEWING OPTIONS WINDOW contains commonly used commands.



The user initiates the commands via mouse pick. The window remains active in the lower left corner of the screen.

A detailed description of each function is given in the following section.

PART (CONTROL)

This command sends the user to the PART CONTROL menu. For a description of the commands in the PART CONTROL menu refer to *PART CONTROL, Section 10.3.*

SAVE (DATABASE)

This command saves and stores all changes to the VPG database (It is recommended that the user save often to avoid the risk of losing work due to system failures, power surges, etc.).

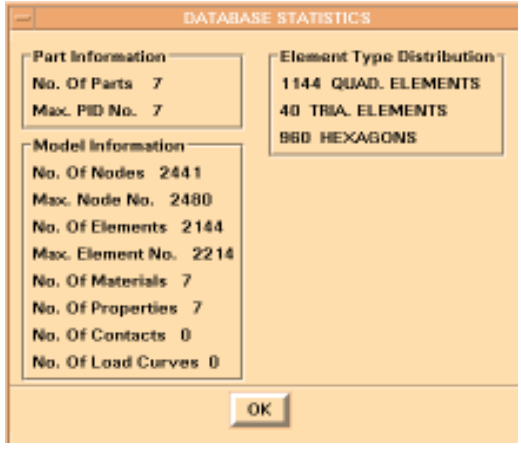
1. VPG prompts:

```
>   FILENAME WRITTEN ON DATE & TIME
>   CREATE A NEW FILE? (YES/NO/ABORT)
?   YES
    NO
    ABORT
```

- YES prompts for a new file name. Once the user enters a new name, the program updates the current file and saves the copy under the new name. The user is automatically relocated to the new file.
- NO updates and saves the current file.
- ABORT exits the function without saving the file.

DATABASE (STAT)ISTICS

This function displays statistics about the VPG database regarding geometry, models, materials, and interfaces. The figures are displayed in the following window:



ACTWIN (ACTIVE WINDOW)

The ACTWIN command allows the user to designate a smaller active viewing window. The command clears everything not captured in the user defined window allowing the user to focus on specific parts of the model. The window is manipulated via the following commands:

```
DIAGONAL NODE/PTS
KEY IN XYZ RANGE
OFF
SCALE ACTIVE WINDOW
DRAG WINDOW
EXIT
```

Clicking on the ACTWIN button after the new window has been defined will turn the window off.

SETUP

This command sends the user to the SETUP menu. For a description of the commands in the SETUP menu refer to *SETUP, Section 10.4.*

UTIL(ITY MENU)

This command sends the user to the UTILITY menu. For a description of the commands in the UTILITY menu refer to *UTILITY, Section 10.5.*

SX - DYNAMIC SCREEN X ROTATION

This shortcut rotates the model about screen X, where the X-axis is parallel to the horizontal edge of the screen. When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

SY - DYNAMIC SCREEN Y ROTATION

This shortcut rotates the model about screen Y, where the Y-axis is parallel to the vertical edge of the screen. When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

SZ - DYNAMIC SCREEN Z ROTATION

This command rotates the model about screen Z, where the Z-axis is perpendicular to the plane of the screen. When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

VX - DYNAMIC VIRTUAL X ROTATION

This command rotates the model about the global X (the X-axis is as displayed in the coordinate system at the lower left side of the screen). When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

VY - DYNAMIC VIRTUAL Y ROTATION

This command rotates the model about the global Y (the Y-axis is as displayed in the coordinate system in the lower left hand corner of the screen). When the user selects this command, VPG displays a ruler bar on the right hand side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

VZ - DYNAMIC VIRTUAL Z ROTATION

This command rotates the model about the global Z (the Z-axis is as displayed in the coordinate system in the lower left corner of the screen). When the user selects this command, VPG displays a ruler bar on the right side of the screen. Moving the cursor up or down causes the model to rotate; tick marks represent 20-degree rotation increments. Clicking the left mouse button stops the rotation.

TOP

This command automatically displays the model from the TOP or XY-plane.

SIDE

This command automatically displays the model from the SIDE or XZ-plane.

REAR

This command automatically displays the model from the REAR or YZ-plane.

FREROT (FREEHAND ROTATION)

The Freehand Rotation (FREROT) command is a combination of SX and SY. Moving the mouse up/down manipulates SX. Moving the mouse left/right manipulates SY. Moving the mouse diagonally combines the movements of both commands. Clicking the left mouse button stops the rotation.

ISOMETRIC

This command automatically displays the model form the ISOMETRIC point of view.

PAN

This command translates the model by following the movement of the cursor. If the cursor is moved off the screen, the cursor reappears at the center of the screen. Clicking the left mouse button stops the pan.

FILL

This command rescales the model to include all entities belonging to parts that are currently turned on. FILL automatically zooms in or out until the model fits the viewing area of the screen.

ZOOM

The Dynamic Zoom (ZO) command displays a cross-hair at the center of the screen and prompts the user to locate a reference point from which to zoom (pick a center). VPG centers the model at that point. The user moves the mouse up or down to zoom in or out respectively.

note: The Cursor Zoom is available in the VIEWING OPTIONS menu.

PRINT

This shortcut creates a Postscript file of the display area and sends the file to the printer (default). Prior to printing, the Postscript driver must be initialized to accommodate the VPG software.

note: The user can change the Printing Defaults in the SETUP/HARDCOPY DEFAULT menu.

PART CONTROL

The functions of the PART CONTROL menu are intended for organizing line, surface, and element data in a structure. A PART is a set of lines, surfaces, and/or elements grouped under a PART NAME. The PART NAME is defined as a label of alpha or numeric data that consists of no more than eight characters. Presently up to 1000 parts may be defined in a database. Each part has a unique Part Identification (PID). The functions in the PART CONTROL menu are organized as follows:

PART CONTROL

ADD ELEMENTS TO PART
ADD LINES TO PART
ADD SURFS TO PART
CONNECTING ELEMENTS
CONNECTING NODES
CHANGE PART COLOR
CREATE
CURRENT
DELETE
KEEP PART
LIST
MASS CALCULATION
ON/OFF
PART COLOR ON/OFF
PART SUMMARY
SEPARATE PARTS
TRANSPARENT MODE
VIEW
MODIFY
EXIT

A detailed description of each function is given in the following sections.

ADD ELEMENTS TO PART

This function allows the user to copy elements from one part to another part. The part to which the elements are added does not have to be current or on.

1. VPG prompts:
   ```
   >  SELECT TARGET PART
   >  PICK A LINE (L) AN ELEM (C) OR PART NAME (P) OF A
      PART.
   ```

2. Once the user selects the target part, VPG prompts:
   ```
   >  ELEMENTS WILL BE ADDED TO PART  xxxx [part name].
   >  SELECT ELEMENTS
   ```

3. The user selects the desired elements, then enters DONE to exit the menu.

ADD LINES TO PART

This function allows the user to copy lines from one part to another part. The part to which the lines are added does not have to be current or on.

1. VPG prompts:
   ```
   >  SELECT TARGET PART
   >  PICK A LINE (L), AN ELEM (C), OR PART NAME (P) OF
      A PART.
   ```

2. Once the user selects the target part, VPG prompts:
   ```
   >  LINES WILL BE ADDED TO PART  xxxx (part name).
   >  SELECT LINES
   ```

3. The user selects the desired lines, then enters DONE to exit the menu.

ADD SURFACES TO PART

This function allows the user to copy surfaces from one part to another part. The part to which the surfaces are added does not have to be current or on.

1. VPG prompts:
   ```
   >  SELECT TARGET PART
   >  PICK A LINE (L), AN ELEM (C), OR PART NAME (P) OF
      A PART.
   ```

2. Once the user selects the target parts, VPG prompts:
   ```
   >  SURFACES WILL BE ADDED TO PART  xxxx [part name].
   >  SELECT SURFACES
   ```

3. The user selects the desired surfaces, then enters DONE to exit the menu.

CONNECTING ELEMENTS

This function is used to determine element connectivity from one part in the model to another.

1. VPG prompts:
   ```
   >  SELECT PART FOR CONNECTING ELEMENTS
   >  PICK AN ELEMENT OR A PART NAME OF A PART
      <part list>
      CURSOR AT ELEMENT
      CURSOR AT LINE
      CURSOR AT SURFACE
      CURRENT PART
      UNDO EXIT
   ```

2. After selecting the desired part, VPG prompts:
   ```
   >  PART XXXXX IS SELECTED
   >  XXXX ELEMENTS FROM XXX PARTS CONNECTED TO XXXX
   >  PICK AN ELEMENT, OR PART NAME TO TURN ON
      <connected part list>
      ALL CONNECTED PARTS
      SELECT BY ELEMENT
   ```

Continued on next page.

UNDO LAST EXIT

- The connecting elements are highlighted.

3. Once the part is selected VPG prompts:
   > PART XXXX IS TURNED ON
   > PICK AN ELEMENT OR PART NAME TO TURN ON
   - This step is repeated until the user selects exit.

CONNECTING NODES

This function is used to determine node connectivity from one part in the model to another.

1. VPG prompts:
   > SELECT PART FOR CONNECTING ELEMENTS
   > PICK AN ELEMENT OR A PART NAME OF A PART
   <part list>
   CURSOR AT ELEMENT
   CURSOR AT LINE
   CURSOR AT SURFACE
   CURRENT PART
   UNDO EXIT

2. After selecting the desired part, VPG prompts:
   > PART XXXXX IS SELECTED
   > XXXX ELEMENTS FROM XXX PARTS CONNECTED TO XXXX
   > PICK AN ELEMENT, OR PART NAME TO TURN ON
   <connected part list>
   ALL CONNECTED PARTS
   SELECT BY ELEMENT
   UNDO LAST EXIT
   - The connecting nodes are highlighted.

3. Once the part is selected VPG prompts:
   > PART XXXX IS TURNED ON
   > PICK AN ELEMENT OR PART NAME TO TURN ON
   - This step is repeated until the user selects exit.

CHANGE PART COLORS

This function allows the user to change the color of a part.

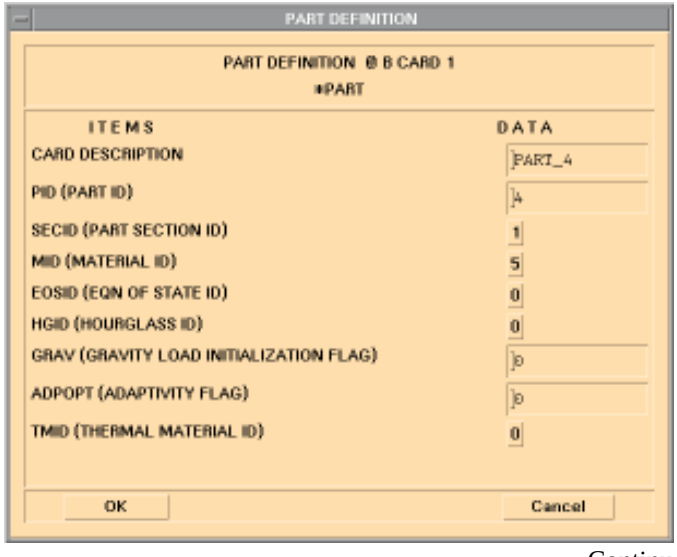
1. VPG prompts:
   ```
   >  SELECT PART TO CHANGE COLOR
   >  PICK A LINE (L), AN ELEM (C), OR PART NAME (P) OF
      A PART.
   ```

2. Once the user selects the target part, VPG prompts:
   ```
   >  PART xxxx (part name) IS SELECTED
   >  SELECT COLOR FROM COLOR BAR OR EXIT
   ```
   - The user selects the desired color from the color bar.

CREATE

This function creates a new part in the VPG database. The user must name the new part with any combination of up to eight, alphanumeric characters.

1. VPG displays the PART DEFINITION window:



Continued on next page.

2. The user edits the window to create the part. If SECID, MID, EOSID, HGID, or TMID is selected the user is prompted to create or edit another window for these properties before retuning to the PART DEFINITION window. OK completes the part and returns the user to the PART CONTROL menu.

note: The new part is displayed as current.

note: The materials and properties of a new part do not have to be defined at the time the part is created. The user may enter OK to bypass step 2, and enter the part's materials and properties at another time.

CURRENT

This function changes the current part. All lines, surfaces, and elements that the user creates are automatically included in the current part. The name of the current part is displayed in its part color in the lower right corner of the screen.

1. VPG prompts:
   > SELECT NEW CURRENT PART
   > PICK A LINE (L), ELEM (C), OR PART NAME (P) OF A PART.
   - The user selects a target part to make current.
   - Once the user selects the target part, VPG makes the part current and displays the part name in the lower right hand corner of the screen.
   - The user can also access the current part function by clicking on the current part name displayed in the DISPLAY PARAMETER OPTIONS WINDOW.

DELETE

This function allows the user to delete a part with its associated surfaces, lines, and elements. The nodes of a part that are selected for deletion become unreferenced and appear as white asterisks ("*") on the screen.

note: Before proceeding further with model generation, the user must ensure that the deleted part is no longer current.

1. VPG prompts:
   > PICK A LINE (L), AN ELEM (C), OR PART NAME (P) OF A PART
   - The user selects a target part .

2. Once the user selects the target, VPG prompts:
   > XXXX LINES, XXXX SURFACES AND XXXX ELEMENTS IN PART YYYY.
   > O.K. TO DELETE PART YYYY?
   ? YES
   NO
   - YES deletes the selected part and its lines, surfaces, and elements.
   - NO deselects the part.

KEEP PART

This function retains the ON status of the selected part(s), and turns OFF the deselected parts. The program automatically re-scales the selected part(s) to fill the screen.

1. VPG prompts:
   > SELECT PART TO KEEP
   > PICK AN ELEMENT, OR PART NAME OF A PART.
   - The user selects a target part(s).
   - VPG highlights the selected line(s) or element(s).

2. Once the user selects the desired part(s), VPG prompts:
   > PART xxxx (part name) IS SELECTED

3. EXIT terminates this function after replotting and re-scaling the selected parts.

MASS CALCULATION

This function determines the mass of a part, group of parts or entire model.

1. VPG prompts:
   > ENTER OUTPUT FILE NAME OR [CR] FOR DEFAULT OR "STOP" FOR QUIT

2. Once the file name is entered VPG prompts:
   > ENTER COMMAND
   ALL PARTS

SELECT PART
EXIT

3. VPG displays the SUMMARY OF MASCAL window with the part(s) properties and displays in the dialogue window the part(s) density.

LIST

This function lists the part names with the corresponding PID numbers and ON/OFF status. Parts that are currently ON are listed in their part color. Parts that are currently OFF are listed in white.

ON/OFF

This function toggles selected parts ON and OFF. The following options are given as a submenu:

1. VPG prompts:
   > PICK AN ELEMENT, OR PART NAME OF A PART.
   PART NAME (clicking on the listed parts will turn them on/off)
   CURSOR AT ELEMENT (select part by element - default)
   CURSOR AT LINE (select part by line)
   CURSOR AT SURFACE (select part by surface)
   INVERSE ON/OFF (all on parts off and all off parts on)
   ON ALL PARTS (all parts are turned on)
   OFF ALL PARTS (all parts are turned off)
   PART SETS ON/OFF (turns parts on/off by their assigned set)
   MULTI PT REGION (select part by user defined multi point region)
   WINDOW (select part by drag window)
   UNDO (rejects last)

PART COLORS ON/OFF (toggle)

This function changes each element type into a separate and uniform color (i.e., quadrilateral elements turn blue; triangular elements turn red; 8-node cube elements turn green; 6-node wedge elements turn magenta; and 4-node tetrahedron elements turn light magenta). This function activates the color changes when selected once, and restores the original colors when selected again.

PART SUMMARY

This function displays statistics regarding the geometry, model, material, and interface information of the selected parts. The figures are displayed in the options area.

1. VPG prompts:
   > SELECT PART FOR SUMMARY
   > PICK AN ELEMENT, OR PART NAME OF A PART.

note: DRAG WINDOW and MULTI-POINT REGION are not valid selections for this command.

2. The following is an example of the statistics display for selected parts:

NO. OF ELEMENTS	108
QUAD ELEMENTS	82
TRI ELEMENTS	26
MINIMUM ELEM. NO.	1472
MAXIMUM ELEM. NO.	1579
MIN NODE NO.	1378
MAX NODE NO.	2249
MATERIAL:	NONE
EL. PROPERTY:	NONE
INTERFACE:	NONE

SEPARATE PART(S)

This function allows the user to separate part(s) from a structure. For example, two parts that are connected by common nodes may be quickly separated (or disconnected) using SEPARATE PART. This function also provides the user with the option to create zero length RBARS between the separated parts. (Common nodes between parts become new nodes once the parts are separated.)

1. VPG prompts:
   > CREATE RIGID LINKS BETWEEN SEPARATED PARTS? (Y/N)
   - YES creates a part called ZEROBAR, in PART CONTROL, that automatically contains all of the newly-created RBARS.

Continued on next page.

2. VPG prompts:
   > SELECT PART OR ALL? (SP/A/E)

3. If the user selects ALL PARTS, VPG prompts:
   > XXXX (# of RBE2s) RBE2S CREATED BETWEEN SEPARATED PARTS
   All modeled parts in the database separate from each other. This includes:
   PLATES
   SOLIDS
   CBARS/CBEAMS
   CELAS1/CELAS2 ELEMENTS
   PLOTEL ELEMENTS
   CROD/CTUBE ELEMENTS
   These entities do not separate:
   RBE2/RBAR ELEMENTS
   CONM2 ELEMENTS
   CORD2R ELEMENTS

4. If the user enters SELECTED PARTS, VPG prompts:
   > SELECT PART TO SEPARATE
   > PICK AN ELEMENT, OR PART NAME OF A PART.
   - The parts separate from each other, and VPG prompts:
     > XXXX (# of RBE2s) RBE2S CREATED BETWEEN SELECTED PARTS.

TRANSPARENT MODE

This function makes selected parts transparent during the shading operation.

1. VPG prompts:
   > PICK A LINE (L), ELEM (C), OR PART NAME (P) OF A PART

2. The user makes the appropriate selection, then enters DONE to exit the menu.

note: The user may adjust the degree of transparency in the SETUP menu (refer to Section 10.4).

MODIFY

Allows user to change a selected parts attributes such as thickness, material etc.

SETUP DEFAULTS

The SETUP DEFAULTS window contains various preprocessing utilities that are toggle-switch activated.

A detailed description of each option is given in the following section.

ANALYSIS PROGRAM

This option adjusts the property and material data to match the selected parameters of the solver. The default solver code for VPG is LS-DYNA.

NASTRAN
DYNA3D (default)

- **MATERIAL PROPERTY** and **ELEMENT PROPERTY** menus change according to the analysis program.

note: VPG is unable to convert existing properties into the selected solver format.

AUTO SEGMENT END (toggle)

This option allows the user to designate the end points of line segments to serve as control points.

note: This function is used in conjunction with CONTROL POINT MESH. VPG prompts the user to select control points on lines that are joined through the line segment option.

CONTROL POINT MESH (toggle)

This function pre-selects the points on a line for the desired node locations associated with the **2L, 3L, 4L, 6S, 8S, 9S,** and **12S** mesh programs.

CORNER BIAS MESH (toggle)

This option allows the user to enter an element bias factor (0.625 to 1.6) relative to the selected corner of the line data that is to be modeled. This function multiplies each adjacent element that is created from the selected corner by the bias factor.

note: A bias factor greater than 1.0 generates proportionately larger elements from the selected corner. A bias factor less than 1.0 generates proportionately smaller elements from the selected corner.

note: This function may be used with only the **2L**, **3L**, **4L**, **6S**, **8S** , **9S**, and **12S** functions.

EDGE BIAS MESH (toggle)

This option allows the user to enter an element bias factor (0.625 to 1.6) relative to the selected edge of the line data that is to be modeled. This function multiplies each adjacent element that is created from the selected edge by the bias factor.

note: A bias factor greater than 1.0 generates proportionately larger elements from the selected edge. A bias factor less than 1.0 generates proportionately smaller elements from the selected edge.

note: This function may be used with only the **2L**, **3L**, **4L**, **6S**, **8S** , **9S**, and **12S** functions.

CHANGE COLOR MAP

This function allows the user to change the default colors of the COLOR BAR for plotting of part color or contour color.

ELEMENT SIZE

This function allows the user to preset a default element size before generating the 2 LINE, 3 LINE, 4 LINE, or SURFACE mesh.

FREE ROTATION

This function allows the user to set the FREE ROTATION function to an arbitrary rotation. The ARBITRARY setting allows the user to return to the model's initial orientation by returning the mouse pointer to the center of the screen (where it started). The default setting for this function is XY rotation. The FREE ROTATION command is a combination of SX and SY. Moving the mouse up/down manipulates SX. Moving the mouse left/right manipulates SY. Moving the mouse diagonally combines the movements of both commands. Clicking the left mouse button stops the rotation.

SHADING OPTIONS

This function offers the following 3 options for shading a part or a model:

GOURAUD SHADING (smooth shading)
FLAT SHADING NO EDGE
FLAT SHADING W/EDGE

Gouraud shading is the default setting for VPG.

TRANSPARENCY FACTOR

This function sets a degree of transparency for the shading option. Factors closer to 0.0 (zero) increase opacity, while factors closer to 1.0 (one) decrease opacity.

ELEMENT TYPE DISPLAY

The toggle switches in the ELEMENT TYPE DISPLAY determine what element types will be shown in the drawing window. The default setting for all the switches is ON.

TYPE IDENTIFIER

This toggle switch controls the display of element type identifiers. If the option is turned on, a symbol for the 1-D element types will be plotted at the center of the element to indicate the element type. The identifier symbols for the various elements are listed in the following table:

ELEMENT TYPE	ANALYSIS PROGRAM	IDENTIFIER SYMBOL
Beam	LS-DYNA & NASTRAN	B
CROD (Truss)	NASTRAN	T
CTUBE (Pipe)	NASTRAN	P
Rigid Element	LS-DYNA & NASTRAN	R
Spotweld	LS-DYNA	W
Discrete	LS-DYNA	D
Seatbelt	LS-DYNA	S
CELAS1 & CELAS2 (Spring)	NASTRAN	S

Default for this option is off.

UTILITY MENU

The commands in the UTILITY menu make up VPG's "tool kit." Many of these commands are also located in other menus.

UTILITY MENU

ANGLE
COORDINATE SYSTEM
DRAW ARROW
DISTANCE
DEFINE TITLE
IDENTIFY NODE/POINT
IDENTIFY ELEMENT
LOAD CURVE
SHOW LINES
HARDCOPY DEFAULTS
EXIT

A detailed description of each function is given in the following sections.

ANGLE

The ANGLE command measures the angle between three selected points and/or nodes. VPG calculates the measurements of four angles for this command. The first is the "true" angle--the angle in the plane of the selected points. The remaining three angles are measured from the TOP, SIDE, and REAR views of the model.

1. VPG prompts:
   > SELECT VERTEX

2. VPG prompts:
   > SELECT 2 POINTS/NODES

3. VPG displays angle information:
   > ANGLE= XX.XX, TOP: XX.XX, SIDE: XX.XX, REAR: XX.XX

4. DONE or EXIT returns the user to the UTILITY menu.

COORDINATE SYSTEM

The functions in this menu create and modify local coordinate systems.

note: The COORDINATE SYSTEM command requires several submenus. To help guide the user, each submenu is described in the following subsections.

1. VPG prompts:
   > ENTER COMMAND
   ? CREATE
   DELETE
   MODIFY
   CURRENT
   DISPLAY ON/OFF
   EXIT

2. After the user enters the appropriate system, VPG lists the options specific to that system.

- COORDINATE SYSTEM
CREATE

This function creates local coordinate systems. The CS command is used in conjunction with the ASSIGN COORDINATE SYSTEM TO NODE command .

1. VPG prompts:
   ```
   >  SELECT LOCAL SYSTEM TYPE
      NODES
      SYSTEM
      VECTOR
   ```

2. VPG prompts the user to create a local system. A thorough explanation of how to create a local coordinate system is covered in *Getting Started, Section 3.5.*

3. After the user enters one of the options above, the local coordinate system appears on screen with its C.S. number.

Depiction of two different coordinate systems.

- COORDINATE SYSTEM
DELETE

This function deletes local coordinate system sets from the database.

1. VPG displays the DEFINE COORDINATE CARD LIST and prompts:
   `> SELECT CARD TO DELETE`

2. EXIT deletes the selected coordinate system from the database and resets the nodes to the global system.

note: To remove the local systems/definitions from individual nodes, the user must reassign the nodes to the global coordinate system (refer to ASSIGN COORDINATE SYSTEM on page 3 of this section).

- COORDINATE SYSTEM
MODIFY

This function modifies the definition of an existing local coordinate system.

1. VPG displays the define coordinate card list and prompts:
   `> EXIT TO CONTINUE`

2. After selecting the coordinate system, VPG displays the CSYS DEFINITION CARD which allows the user to select and edit the coordinate systems data.

3. Once the user accepts the modified card, the new local system appears on the screen. All local systems belonging to the modified coordinate system automatically adjust to match the labeled coordinate system.

- COORDINATE SYSTEM
CURRENT

This function switches the current coordinate system form local to global.

- COORDINATE SYSTEM
DISPLAY ON/OFF

This function toggles the local coordinate systems on or off.

DRAW ARROW

This function allows the user to draw arrows about a specific image on the display screen.

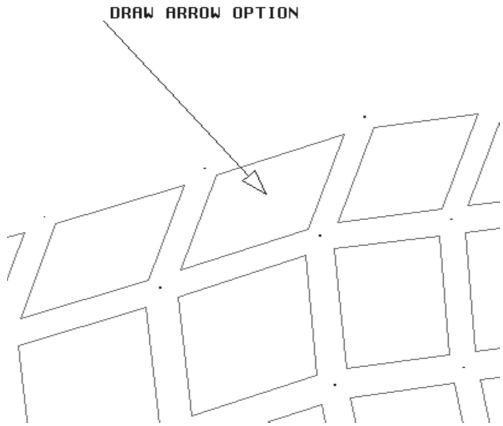
note: This command is used in conjunction with DEFINE TITLE.

1. VPG prompts:
   ```
   >   PICK THE TAIL LOCATION FOR THE ARROW
   ?   CURSOR LOCATION
       EXIT
       UNDO LAST
   ```

2. After the user selects the location of the tail, VPG prompts for the location of the head.
   ```
   >   PICK THE HEAD LOCATION FOR THE ARROW
   ?   CURSOR LOCATION
       EXIT
       UNDO LAST
   ```

3. After the user defines the location of the head, VPG draws the arrow and returns to step 2.

Continued on next page.



DISTANCE

This function calculates the distance between 2 points, 2 nodes, or a node and a point. This command is also listed in the ELEMENT OPTIONS and NODE OPTIONS menus.

1. VPG prompts the user to select a node or point:
   > SELECT TWO POINTS/NODES
   - After the user selects the appropriate points or nodes, VPG displays in the prompt area of the screen the vector distance and the DX, DY, and DZ components.

2. ABORT, DONE, or EXIT terminates this function.

DEFINE TITLE

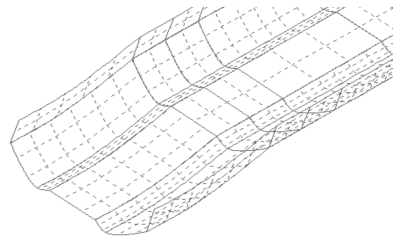
This function allows the user to enter a title or a text at any location on the display screen.

1. VPG prompts:
   > INPUT CHARACTER SIZE (1.0 - 10.0) <1.0>

2. After the user selects a font size, VPG prompts:
   > DEFINE TITLE STARTING LOCATION
   - The user defines the location of the text by positioning the cursor at the desired point on the screen and clicking the left mouse button.

3. VPG prompts:
   > ENTER TITLE
   - The user types in the text and presses the enter key. VPG displays the text at the selected location.

BUMPER1



IDENTIFY NODES/POINTS

This function identifies any node and its corresponding global location in X, Y, Z coordinates.

1. VPG prompts:
   > PICK NODES/POINTS
   - If the user selects ALL NODES, VPG displays the numbers of the nodes at their corresponding node locations. The user returns to the UTILITY menu.
   - If the user selects NODE, VPG displays the number of the node that is closest in proximity to the cursor. VPG issues the following message on the command line:
     > NODE xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - If the user selects POINT, VPG displays the number of the point that is closest in proximity to the cursor. VPG issues the following message on the command line:
     > POINT xxxx X = x.xxx Y = y.yyy Z = z.zzz
   - DONE or EXIT returns the user to the UTILITY MENU.

IDENTIFY ELEMENTS

This function allows the user to identify an element and its nodes by cursor selection.

1. VPG prompts:
   > SELECT ELEMENT BY CURSOR
   - VPG identifies the element's respective type, number, part and nodes. The selected element will be highlighted, and element and node numbers will be displayed in the dialogue window.
   - The user also has the optoin to key in the number of a desired element to identify its location and characteristics.
   - It will also identify:

NASTRAN ELEMENT TYPE	DYNA ELEMENT TYPE
CBAR	BEAM
RBE2	RIGID EL
PLOTEL	PLOTEL
PLATE	PLATE
SOLID	SOLID
CELAS2	SPRING
CELAS1	SPOTWELD
RBAR	MASS
CTUBE	JOINT
ISO-SHL	T-SHELL
RBE3	SEATBELT

IDENTIFY ELEMENTS displays element numbers and corresponding node numbers.

LOAD CURVE

This function defines and modifies the load curves for a nonlinear analysis.
The LOAD CURVE command requires several levels of submenus. Each command level is fully explained in their respective sections of the *User's Manual*.

1. VPG prompts:
   ```
   >   ENTER COMMAND
   ?   CREATE
       DELETE
       LIST
       MODIFY
       READ CURVE
       SHOW CURVE
       EXIT
   ```

2. After the user selects a command, VPG lists a set of options specific to that command.

- LOAD CURVE
CREATE

This function allows the user to create load curves.

1. VPG displays the CURVE DEFINITION CARD allowing the user to define the load curve.
   ```
   >   ENTER LOAD CURVE NUMBER ( xxxx) OR -1 TO EXIT
   >   ENTER CURVE NAME (UP TO 8 CHARACTERS)
   ```
   - VPG recommends using the extension **.lcr** for this command.

2. VPG prompts:
   ```
   >   ENTER DATA (TIME & VALUE) FOR POINT  XXXX OR END
   ```
   - Where XXXX is the point number and the input format is TIME, VALUE.

3. VPG displays the curve operations:
   ```
   ?   CHANGE GRAPH LOCATION
   ```

Continued on next page.

```
CURVE OPERATIONS
CLEAR SCREEN
DELETE CURVES
HARDCOPY
OPTIONS
QUERY
RANGE
SAVE CURVES
EXIT
```

- These options are explained in *GRAPH, Section 8.4.*

- LOAD CURVE
DELETE

This function allows the user to delete load curves.

1. VPG displays the DEFINE CURVE CARD LIST, and prompts:
   > SELECT CARD TO DELETE

2. EXIT deletes the selected load curves from the database.

- LOAD CURVE
LIST

This function lists the available load curves in the DEFINE CURVE CARD LIST.

- LOAD CURVE
MODIFY

This function allows the user to quickly modify points on a defined curve.

1. VPG lists the curves in the DEFINE CURVE CARD LIST, and prompts:
   > SELECT CURVE FROM CURVE LIST

2. After selecting the curve, the CURVES DEFINITION CARD appears allowing the user to edit the curve data.

- LOAD CURVE
READ CURVE

This function imports existing curve data into the VPG database.

1. VPG displays the VPG FILE MENU allowing the user to enter the curve (**.cur**) file.

2. Once the curve file is selected and the data is read in, VPG displays the DEFINE CURVE CARD LIST. The user has the choice of selecting the curves before they are plotted.

note: The load curve database format is shown below. ALL text must be entered with CAPITAL letters.

LOAD CURVE FILE FORMAT

line 1:	$ VPG LOAD CURVE (Header line)
line 2:	CURVE, CURVE ID, TYPE, CURVE TITLE (A5, I5, I5, 1X, A8)
line 3:	X1, Y1 (2F10.4)
line 4:	X2, Y2 (2F10.4)
line 5:	X3, Y3 (2F10.4)
	etc.
line n:	CURVE, CURVE ID, TYPE, CURVE TITLE (Repeat a new curve)
line n+1:	X1, Y1 (2F10.4)
line n+2:	X2, Y2 (2F10.4)
line n+3:	X3, Y3 (2F10.4)
	etc.
last line:	END (A3) (Terminate Input)

Where:

CURVE:	Header (Required).
CURVE ID:	Load Curve Number
TYPE:	Curve Type (0- Transient Analysis; 1- Stress Initialization)
CURVE TITLE:	Curve's Title (User Defined)
X1, Y1:	Curve Point 1
X2, Y2:	Curve Point 2
END:	Header (Required)

Continued on next page.

The following is an example of the LOAD CURVE file:

```
$ VPG LOAD CURVE
CURVE    2    0 CURVE  2
    0.0000    0.0000
    1.0000    1.5000
    2.0000   -0.0000
    3.0000    0.5000
    4.0000   -1.5000
CURVE    4    0 CURVE  4
    0.0000    2.0000
    6.0000   -3.5000
    7.0000    1.5000
    8.0000    2.0000
END
```

- LOAD CURVE

SHOW CURVE

This function displays load curves.

1. VPG displays the DEFINE CURVE CARD LIST.

2. After selecting a curve, VPG displays the following curve operations:

```
?   CURVE OPERATIONS
    CLEAR SCREEN
    DELETE
    HARDCOPY
    OPTIONS
    QUERY
    RANGE
    SAVE
    EXIT
```

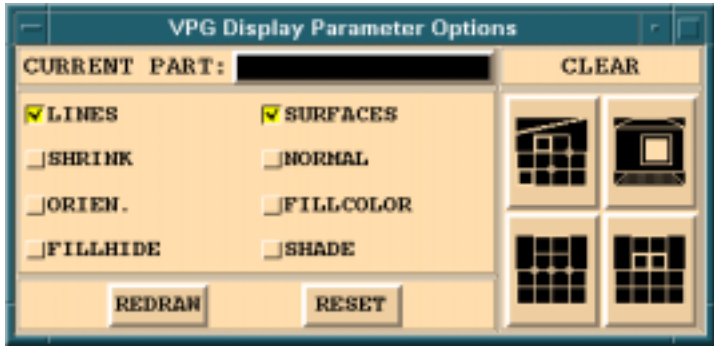
   - These options are explained in *GRAPH, Section 8.4.*

SHOW LINES

This function identifies any existing line and its direction. VPG highlights the selected line and marks its points with arrowheads that are oriented in the direction of the line.

DISPLAY PARAMETER OPTIONS WINDOW

The functions of the DISPLAY PARAMETER OPTIONS WINDOW control various aspects of the model's appearance.



The user initiates the commands via mouse pick. The window remains active in the lower right corner of the screen.

A detailed description of each function is given in the following pages.

LINES — Turns plot lines on/off.

SHRINK — This function reduces the size of all elements by 20 percent. SHRINK is also useful for locating any missing elements in a shell or solid structure.

ORIEN. — This function displays element orientation. Beam element orientation is shown by drawing a vector at the first node of the beam toward the element local Y-axis. Shell orientation is shown by drawing an arrow from the first node of an element toward the second node of an element. Solid element orientation is shown by drawing an arrow from the bottom surface toward the top surface. This display is useful for picturing the beam orientation and the normal direction of the shell elements (the normal of the shell follows the right hand rule from the first node to the second node).

note: The clarity of the element orientation display improves when the SHRINK ELEM ON/OFF command is activated.

FILLHIDE — This function creates a hidden surface plot for all of the displayed elements, which provides the user with a realistic view of the model.

SURFACES — Turns highlighted surfaces on/off.

NORMAL — This function displays the orientation of an element with an arrow that is at the centroid of the element and that is perpendicular to the surface of the element. For a solid element, the arrow points toward the bottom surface of the element.

SHADE — This command displays the elements as if they were illuminated by a light source. Elements that are not directly exposed to the light source are appropriately “shaded” to imitate actual shading.

FILL COLOR — The FILLCOLOR option fills the displayed elements with a designated part color.

REDRAW — VPG is currently designed to update the screen after each command. Occasionally, the use of specific commands requires that images be updated with an additional step (e.g., when the user operates the dynamic zoom at the same time that the element normals are displayed, he or she must activate the REDRAW command to resize the arrows that represent the element normals).

RESET — Returns the DISPLAY PARAMETER OPTIONS WINDOW to its default settings.

CLEAR — The clear buttons in the DISPLAY PARAMETER OPTIONS WINDOW clear highlighted elements, nodes, lines, etc. The buttons are highlighted and tooltips are available when the functions are selectable.

This function removes highlights from the screen such as those that are made with the commands SHOW LINE, BOUNDARY CHECK, ID ELEMENTS, etc. Entities that are highlighted remain so until this button is activated.

This function removes the active window and returns the user to the display window.

This function removes the highlights from nodes after the CONNECTING NODES function has been used.

This function removes the highlights from elements after the CONNECTING ELEMENTS function has been used.

CURRENT PART — The current part name window not only lists the current part, it also serves as a short cut to execute the current part command. Clicking on this window will execute the CURRENT PART command allowing the user to set a new current part.

Appendix A
VPG Capabilities for F.E.A. Analysis Programs

	NASTRAN	LS-DYNA
ELEMENT TYPE		
Beam	Y	Y
Truss	Y	Y
Circular Tube	Y	Y
PLOTEL	Y	N/A
Runner (Hot, Cold)	N/A	N/A
Plate (Quad., Tri.)	Y	Y
Iso-parametric Shell (Quad8, Tria6)	Y	N/A
Solid (Cube, Wedge, Tetra)	Y	Y
Concentrated Mass	Y	Y
Spring	Y	Y
Rigid Link	Y	Y
MATERIAL		
Elastic	Y	Y
Non-linear (number of types)	N	All
BOUNDARY CONDITION		
SPC	Y	Y
MPC	N	Y
Polymer Entrance	N/A	N/A
LOADING		
Nodal Force	Y	Y
Nodal Moment	Y	Y
Pressure Load	Y	Y
Initial Velocity	Y	Y
CONTACT INTERFACE		
Sliding Interface	N/A	Y
Stonewall	N/A	Y
Master/Slave Segment	N/A	Y
Slave Node	N/A	Y

Continued on next page.

	NASTRAN	LS-DYNA
LOCAL COORDINATES		
Rectangular	Y	Y
Cylindrical	Y	Y
Spherical	Y	Y
LOAD CURVE		
Load Curve	N/A	Y
POST PROCESSING		
Deformed Shape	Y	Y
Mode Shape Plot	Y	Y
Contour Line/Fill Plot	Y	Y
Stress Contour	Y	Y
Strain Contour	Y	Y
Deformed Contour	Y	Y
Strain Energy Contour	Y	Y
Deformed Animation	Y	Y
Mode Shape Animation	Y	N/A
Frequency Response	Y	N/A
Stress Contour Animation	Y	Y
Deform-Contour Animation	Y	Y
Forming Limit Diagram	N/A	Y
Mold Filling Animation	N/A	N/A
X-Y PLOT		
Time History (Displacement, Stress, Energy)	Y	Y
X-Y Plot Between Variables	Y	Y
Frequency Response Curve	Y	N/A
Curve Operation (Add, subtract, etc.)	Y	Y
Curve Filter	Y	Y

Appendix B
VPG Hardware and Software Requirements

PLATFORM	OS	VERSION	GRAPHICS BOARD	MINIMUM MEMORY	GRAPHIC LIBRARY
HP	HP-UX	9.01+	Minimum CRX 8 Plane	128MB	PHIGS
IBM	AIX	3.2+	Minimum 24 plane graphics	128MB	PHIGS
SGI	IRIX	5.3+	All graphhics boards supported	128MB	N/A
SUN	SOLARIS	2.3+	MinimumGX+	128MB	PHIGS
DEC	OSF1	V3.0	Minimum PXG+	128MB	PHIGS
	OSF1	V4.0	Minimum PXG+	128MB	PHIGS

Appendix C
Supported IGES Entity Types

NAME	TYPE
Null Entity	0
Circular Arc Entity	100
Composit Curve Entity	102
Conic Arc Entity	104
Copious Data Entity	106
Plane Entity	108
Line Entity	110
Parametric Spline Curve Entity	112
Parametric Spline Surface Entity	114
Point Entity	116
Ruled Surface Entity	118
Surface of Revolution Entity	120
Tabulated Cylinder Entity	122
Transformation Matrix Entity	124
Rational B-Spline Curve Entity	126
Rational B-Spline Surface Entity	128
Offset Surface Entity	140
Boundary Entity	141
Curve on a Parametric Surface Entity	142
Bounded Surface Entity	143
Trimmed (Parametric) Surface Entity	144
Subfigure Definition Entity	308
Associativity Instance Entity	402
Property Entity	406
Singular Subfigure Instance Entity	408

Appendix D
Converting RADIOSS to LS-DYNA

Element, property and condition equivalancies are displayed below for RADIOSS to LS-DYNA conversion.

A. ELEMENT:

RADIOSS(V3.1) ===>	LS-DYNA(V940)
BEAM	BEAM
TRUSS	BEAM
SOLID	SOLID
SHELL	SHELL
SPRING	DISCRETE (lost the third node.)
MASS	MASS
JOINT	CONSTRAINED_NODAL_RIGID_BODY
RIGID BODY	CONSTRAINED_NODAL_RIGID_BODY
SPOTWELD	CONSTRAINED_SPOTWELD

B. ELEMENT PROPERTY:

Delete VOID, RIVET, AIRBAG property.
Put spring stiffness into DYNA3D's material.

RADIOSS(V3.1) ===>	LS-DYNA(V940)
1-SHELL	SECTION_SHELL
2-TRUSS	SECTION_BEAM(truss)
3-BEAM	SECTION_BEAM(resultant)
4-SPRING	SECTION_DISCRETE
6-ORTHOTROPIC SOLID	SECTION_SOLID
8-GENERAL SPRING	SECTION_DISCRETE
9-ORTHOTROPIC SHELL	SECTION_SHELL
10-COMP. SHELL	SECTION_SHELL
11-COMP. SHELL	SECTION_SHELL
12-3-NODES SPRING	SECTION_DISCRETE
13-BEAM TYPE SPRING	SECTION_DISCRETE
14-GENERAL SOLID	SECTION_SOLID

C. MATERIAL PROPERTY:

RADIOSS(V3.1) ===>	LS-DYNA(V940)
0 -VOID	1 -MAT_ELASTIC
1 -ELASTIC	1 -MAT_ELASTIC
2 -ELASTIC_PLASTIC	12-MAT_ISOTROPIC_ELASTIC_PLASTIC
3 -ELASTIC_PLASTIC_HYDRODYNAMIC	10-MAT_ELASTIC_PLASTIC_HYDRO
4 -JOHNSON_COOK	15-MAT_JOHNSON_COOK
6 -HYDRODYNAMIC_VISCOUS	9 -MAT_NULL
10-ELASTIC_PLASTIC_DP	78-MAT_SOIL_CONCRETE
14-ELASTIC_PLASTIC_ORTHOTROPIC	2 -MAT_ORTHOTROPIC_ELASTIC
19-ELASTIC_ORTHOTROPIC	2 -MAT_ORTHOTROPIC_ELASTIC
21-ELASTIC_PLASTIC_DP	78-MAT_SOIL_CONCRETE
22-ELASTIC_PLASTIC	81-MAT_PLASTICITY_WITH_DAMAGE
23-ELASTIC_PLASTIC	81-MAT_PLASTICITY_WITH_DAMAGE
24-ELASTIC_PLASTIC_BRITTLE	16-MAT_PSEUDO_TENSOR
25-ELASTIC_PLASTIC_ORTHOTROPIC	2 -MAT_ORTHOTROPIC_ELASTIC
27-ELASTIC_PLASTIC_BRITTLE	81-MAT_PLASTICITY_WITH_DAMAGE
28-ORTHOTROPIC	2 -MAT_ORTHOTROPIC_ELASTIC
32-ELASTIC_PLASTIC_ORTHOTROPIC	3 -MAT_PLASTIC_KINEMATIC
33-VISCOPLASTIC	53-MAT_CLOSED_CELL_FOAM
34-VISCOELASTIC	61-MAT_KELVIN-MAXWELL_VISCOELASTIC
35-VISCOELASTIC	76-MAT_GENERAL_VISCOELASTIC
36-ELASTIC_PLASTIC	24-MAT_PIECEWISE_LINEAR_PLASTICITY
38-VISCOELASTIC	76-MAT_GENERAL_VISCOELASTIC

D. BOUNDARY AND LOAD CONDITION:

RADIOSS(V3.1) ===>	LS-DYNA(V940)
SPC	BOUNDARY_SPC_SET
CONCENTRATED LOAD	LOAD_NODE_SET
PRESSURE LOAD	LOAD_SEGMENT_SET
INITIAL VELOCITY	INITIAL_VELOCITY
FIXED VELOCITY	BOUNDARY_PRESCRIBED_MOTION_SET

E. CONTACT AND RIGIDWALL:

Delete interface type 4, including 0 slave id.
Convert slave/master shell material/property set into segment set.

RADIOSS(V3.1) ===>	LS-DYNA(V940)
2 -TIED	CONTACT_TIED_SURFACE_TO_SURFACE
3 -SLIDE/VOID	CONTACT_SURFACE_TO_SURFACE
4 -SLIDE/VOID	CONTACT_SINGLE_SURFACE
5 -SLIDE/VOID	CONTACT_ONE_WAY_SURFACE_TO_SURFACE
6 -SLIDE/VOID	CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY
7 -SLIDE/VOID	CONTACT_SURFACE_TO_SURFACE
8 -SLIDE	CONTACT_DRAWBEAD
10-TIED/VOID	CONTACT_TIED_SURFACE_TO_SURFACE
RIGIDWALL	RIGIDWALL_GEOMETRIC

F. OTHERS:

Delete TRUSS SET, MAT SET, PROP SET.
Delete ACCELEROMETER, MONITORED VOLUME, SECTIONS, SENSOR, CONTROL CARDS.

RADIOSS(V3.1) ===>	LS-DYNA(V940)
FUNCTION	DEFINE_CURVE
SKEW(moving)	DEFINE_COORDINATE_NODES
SKEW(fixed)	DEFINE_COORDINATE_VECTOR

Appendix E
Converting LS-DYNA to RADIOSS

Element, property and condition equivalancies are displayed below for LS-DYNA to RADIOSS conversion.

A. ELEMENT:

Delete SEATBELT elements, INERTIA elements.

LS-DYNA(V940)	===>	RADIOSS(V3.1)
BEAM		BEAM
SOLID		SOLID
SHELL		SHELL
T-SHELL		SOLID
DISCRETE		SPRING (lost the VID.)
MASS		MASS
JOINT		JOINT
CONSTRAINED_NODAL_RIGID_BODY		RIGID BODY
CONSTRAINED_NODE_SET		RIGID BODY
CONSTRAINED_SPOTWELD		SPOTWELD

B. ELEMENT PROPERTY:

Delete SEATBELT section.
Put spring stiffness into RADIOSS's element property.

LS-DYNA(V940)	===>	RADIOSS(V3.1)
SECTION_BEAM(integrated)		3 -BEAM
SECTION_BEAM(resultant)		3 -BEAM
SECTION_BEAM(discrete)		3 -BEAM
SECTION_BEAM(truss)		3 -BEAM
SECTION_BEAM(spotweld)		3 -BEAM
SECTION_DISCRETE		4 -SPRING
SECTION_SHELL		1 -SHELL
SECTION_SOLID		14-GENERAL SOLID
SECTION_SOLID_ALE		14-GENERAL SOLID
SECTION_TSHELL		14-GENERAL SOLID

C. MATERIAL PROPERTY:

LS-DYNA(V940) ===>	RADIOSS(V3.1)
1 -MAT_ELASTIC	1 -ELASTIC
1 -MAT_ELASTIC_FLUID	1 -ELASTIC
2 -MAT_ORTHOTROPIC_ELASTIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
2 -MAT_ANISOTROPIC_ELASTIC	1 -ELASTIC
3 -MAT_PLASTIC_KINEMATIC	32-ELASTIC_PLASTIC_ORTHOTROPIC
4 -MAT_ELASTIC_PLASTIC_THERMAL	2 -ELASTIC_PLASTIC
5 -MAT_SOIL_AND_FOAM	33-VISCOPLASTIC
6 -MAT_VISCOELASTIC	34-VISCOELASTIC
7 -MAT_BLATZ-KO_RUBBER	1 -ELASTIC
8 -MAT_HIGH_EXPLOSIVE_BURN	1 -ELASTIC
9 -MAT_NULL	6 -HYDRODYNAMIC_VISCOUS
10-MAT_ELASTIC_PLASTIC_HYDRO	3 -ELASTIC_PLASTIC_HYDRODYNAMIC
11-MAT_STEINBERG	1 -ELASTIC
12-MAT_ISOTROPIC_ELASTIC_PLASTIC	2 -ELASTIC_PLASTIC
13-MAT_ISOTROPIC_ELASTIC_FAILURE	2 -ELASTIC_PLASTIC
14-MAT_SOIL_AND_FOAM_FAILURE	33-VISCOPLASTIC
15-MAT_JOHNSON_COOK	4 -JOHNSON_COOK
16-MAT_PSEUDO_TENSOR	24-ELASTIC_PLASTIC_BRITTLE
17-MAT_ORIENTED_CRACK	2 -ELASTIC_PLASTIC
18-MAT_POWER_LAW_PLASTICITY	2 -ELASTIC_PLASTIC
19-MAT_STRAIN_RATE_DEPENDENT_PLASTICITY	2 -ELASTIC_PLASTIC
20-MAT_RIGID	1 -ELASTIC
21-MAT_ORTHOTROPIC_THERMAL	14-ELASTIC_PLASTIC_ORTHOTROPIC
22-MAT_COMPOSITE_DAMAGE	14-ELASTIC_PLASTIC_ORTHOTROPIC
23-MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
24-MAT_PIECEWISE_LINEAR_PLASTICITY	36-ELASTIC_PLASTIC
25-MAT_GEOLOGICAL_CAP_MODEL	24-ELASTIC_PLASTIC_BRITTLE
26-MAT_HONEYCOMB	2 -ELASTIC_PLASTIC
27-MAT_MOONEY-RIVLIN_RUBBER	1 -ELASTIC
28-MAT_RESULTANT_PLASTICITY	2 -ELASTIC_PLASTIC
29-MAT_FORCE_LIMITED	2 -ELASTIC_PLASTIC

30-MAT_CLOSED_FORM_SHELL_PLASTICITY
2-ELASTIC_PLASTIC
31-MAT_FRAZER_NASH_RUBBER_MODEL 1-ELASTIC
32-MAT_LAMINATED_GLASS 2-ELASTIC_PLASTIC
33-MAT_BARLAT_ANISOTROPIC_PLASTICITY
2-ELASTIC_PLASTIC
34-MAT_FABRIC 14-ELASTIC_PLASTIC_ORTHOTROPIC
35-MAT_PLASTIC_GREEN-NAGHDI_RATE 32-ELASTIC_PLASTIC_ORTHOTROPIC
36-MAT_3-PARAMETER_BARLAT 2-ELASTIC_PLASTIC
37-MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC
2-ELASTIC_PLASTIC
38-MAT_BLATZ-KO_FOAM 33-VISCOPLASTIC
39-MAT_FLD_TRANSVERSELY_ANISOTROPIC
2-ELASTIC_PLASTIC
40-MAT_NONLINEAR_ORTHOTROPIC 14-ELASTIC_PLASTIC_ORTHOTROPIC
41-MAT_USER_DEFINED_MATERIAL_MODELS
1-ELASTIC
51-MAT_BAMMAN 2-ELASTIC_PLASTIC
52-MAT_BAMMAN_DAMAGE 2-ELASTIC_PLASTIC
53-MAT_CLOSED_CELL_FOAM 33-VISCOPLASTIC
54-MAT_ENHANCED_COMPOSITE_DAMAGE
14-ELASTIC_PLASTIC_ORTHOTROPIC
57-MAT_LOW_DENSITY_FOAM 33-VISCOPLASTIC
59-MAT_COMPOSITE_FAILURE_SHELL_MODEL
14-ELASTIC_PLASTIC_ORTHOTROPIC
59-MAT_COMPOSITE_FAILURE_SOLID_MODEL
14-ELASTIC_PLASTIC_ORTHOTROPIC
60-MAT_ELASTIC_WITH_VISCOSITY 1-ELASTIC
61-MAT_KELVIN-MAXWELL_VISCOELASTIC
34-VISCOELASTIC
62-MAT_VISCOUS_FOAM 33-VISCOPLASTIC
63-MAT_CRUSHABLE_FOAM 33-VISCOPLASTIC
64-MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY
2-ELASTIC_PLASTIC
65-MAT_MODIFIED_ZERILLI_ARMSTRONG 2-ELASTIC_PLASTIC
66-MAT_LINEAR_ELASTIC_DISCRETE_BEAM
1-ELASTIC

67-MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM	1 -ELASTIC
68-MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM	2 -ELASTIC_PLASTIC
69-MAT_SID_DAMPER_DISCRETE_BEAM	1 -ELASTIC
70-MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM	1 -ELASTIC
71-MAT_CABLE_DISCRETE_BEAM	1 -ELASTIC
72-MAT_CONCRETE_DAMAGE	24-ELASTIC_PLASTIC_BRITTLE
73-MAT_LOW_DENSITY_VISCOUS_FOAM	33-VISCOPLASTIC
75-MAT_BILKHU/DUBOIS_FOAM	33-VISCOPLASTIC
76-MAT_GENERAL_VISCOELASTIC	35-VISCOELASTIC
77-MAT_HYPERELASTIC_RUBBER	1 -ELASTIC
77-MAT_OGDEN_RUBBER	1 -ELASTIC
78-MAT_SOIL_CONCRETE	10-ELASTIC_PLASTIC_DP
79-MAT_HYSTERETIC_SOIL	24-ELASTIC_PLASTIC_BRITTLE
81-MAT_PLASTICITY_WITH_DAMAGE	22-ELASTIC_PLASTIC
83-MAT_FU_CHANG_FOAM	33-VISCOPLASTIC
86-MAT_ORTHOTROPIC_VISCOELASTIC	14-ELASTIC_PLASTIC_ORTHOTROPIC
87-MAT_CELLULAR_RUBBER	1 -ELASTIC
88-MAT_MTS	1 -ELASTIC
90-MAT_ACOUSTIC	1 -ELASTIC
96-MAT_BRITTLE_DAMAGE	24-ELASTIC_PLASTIC_BRITTLE
100-MAT_SPOTWELD	1 -ELASTIC
103-MAT_ANISOTROPIC_VISCOPLASTIC	33-VISCOPLASTIC
126-MAT_MODIFIED_HONEYCOMB	2 -ELASTIC_PLASTIC
OTHER MATERIAL	1 -ELASTIC

D. BOUNDARY AND LOAD CONDITION:

LS-DYNA(V940) ===>	RADIOSS(V3.1)
BOUNDARY_SPC_NODE	SPC
BOUNDARY_SPC_SET	SPC
INITIAL_VELOCITY	INITIAL VELOCITY
INITIAL_VELOCITY_NODE	INITIAL VELOCITY
INITIAL_VELOCITY_GENERATION	INITIAL VELOCITY

LOAD_NODE_POINT	CONCENTRATED LOAD (delete it, when DOF is equal to 4 or 8)
LOAD_NODE_SET	CONCENTRATED LOAD (delete it, when DOF is equal to 4 or 8)
LOAD_SEGMENT	PRESSURE LOAD
LOAD_SEGMENT_SET	PRESSURE LOAD
LOAD_SHELL_ELEMENT	PRESSURE LOAD
LOAD_SHELL_SET	PRESSURE LOAD
BOUNDARY_PRESCRIBED_MOTION_NODE(velocity)	FIXED VELOCITY
BOUNDARY_PRESCRIBED_MOTION_RIGID(velocity)	FIXED VELOCITY
BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL(velocity)	FIXED VELOCITY
BOUNDARY_PRESCRIBED_MOTION_SET(velocity)	FIXED VELOCITY

E. CONTACT AND RIGIDWALL:

Delete CONTACT_ENTITY, CONTACT_INTERIOR, CONTACT_1D,
CONTACT_2D_AUTOMATIC, CONTACT_2D_PENALTY,
CONTACT_2D_PENALTY_FRICTION, CONTACT_2D_SINGLE_SURFACE,
CONTACT_2D_SLIDING_ONLY, CONTACT_2D_SLIDING_VOIDS,
CONTACT_2D_TIED_SLIDING.
Convert slave type 2,3,5,6 into segment set.
Convert master type 2,3 into segment set.

LS-DYNA(V940)	===>	RADIOSS(V3.1)
CONTACT_AIRBAG_SINGLE_SURFACE		4-SLIDE/VOID
CONTACT_AUTOMATIC_GENERAL		3-SLIDE/VOID
CONTACT_AUTOMATIC_NODES_TO_SURFACE		7-SLIDE/VOID
CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE		5-SLIDE/VOID
CONTACT_AUTOMATIC_SINGLE_SURFACE		4-SLIDE/VOID
CONTACT_AUTOMATIC_SURFACE_TO_SURFACE		3-SLIDE/VOID
CONTACT_CONSTRAINT_NODES_TO_SURFACE		7-SLIDE/VOID
CONTACT_CONSTRAINT_SURFACE_TO_SURFACE		7-SLIDE/VOID
CONTACT_DRAWBEAD		8-SLIDE
CONTACT_ERODING_NODES_TO_SURFACE		7-SLIDE/VOID
CONTACT_ERODING_SINGLE_SURFACE		4-SLIDE/VOID
CONTACT_ERODING_SURFACE_TO_SURFACE		7-SLIDE/VOID

CONTACT_FORCE_TRANSDUCER_PENALTY	7-SLIDE/VOID
CONTACT_FORCE_TRANSDUCER_CONSTRAINT	7-SLIDE/VOID
CONTACT_FORMING_NODES_TO_SURFACE	7-SLIDE/VOID
CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE	5-SLIDE/VOID
CONTACT_FORMING_SURFACE_TO_SURFACE	3-SLIDE/VOID
CONTACT_NODES_TO_SURFACE	7-SLIDE/VOID
CONTACT_ONE_WAY_SURFACE_TO_SURFACE	5-SLIDE/VOID
CONTACT_RIGID_NODES_TO_RIGID_BODY	7-SLIDE/VOID
CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY	6-SLIDE/VOID
CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY	6-SLIDE/VOID
CONTACT_SINGLE_EDGE	7-SLIDE/VOID
CONTACT_SINGLE_SURFACE	4-SLIDE/VOID
CONTACT_SLIDING_ONLY	3-SLIDE/VOID
CONTACT_SLIDING_ONLY_PENALTY	3-SLIDE/VOID
CONTACT_SURFACE_TO_SURFACE	3-SLIDE/VOID
CONTACT_TIEBREAK_NODES_TO_SURFACE	7-SLIDE/VOID
CONTACT_TIEBREAK_SURFACE_TO_SURFACE	7-SLIDE/VOID
CONTACT_TIED_NODES_TO_SURFACE	7-SLIDE/VOID
CONTACT_TIED_SHELL_EDGE_TO_SURFACE	7-SLIDE/VOID
CONTACT_TIED_SURFACE_TO_SURFACE	2-TIED
RIGIDWALL	RIGIDWALL

F. OTHERS:

Delete T-SHELL SET.
Delete all others which can’t be converted.

LS-DYNA(V940) ===>	RADIOSS(V3.1)
DEFINE_COORDINATE_NODES	SKEW_MOV
DEFINE_COORDINATE_SYSTEM	SKEW_MOV
DEFINE_COORDINATE_VECTOR	SKEW_FIX
DEFINE_CURVE	FUNCTION

Appendix F
Converting NASTRAN to LS-DYNA

Element, property and condition equivalancies are displayed below for NASTRAN to LS-DYNA conversion.

A. ELEMENT:

When RBE2 elements share a common node, merge them into one RBE2.
Delete RBE3 elements, CGAP elements.

NASTRAN	===>	LS-DYNA(V940)
CBAR		BEAM
RBE2		CONSTRAINED_NODAL_RIGID_BODY, CONSTRAINED_NODE_SET (according to it's DOF)
SHELL		SHELL
SOLID		SOLID
CELAS2		DISCRETE (create DEFINE_SD_ORIENTATION)
CELAS1		DISCRETE (create DEFINE_SD_ORIENTATION)
RBAR		CONSTRAINED_SPOTWELD
CONM2		MASS & INERTIA
CBEAM		BEAM
CROD		BEAM
CTUBE		BEAM
ISOPARAMETRIC SHELL		SHELL

B. ELEMENT PROPERTY:

Delete PGAP property.
Put spring stiffness into LS-DYNA's material.

NASTRAN	===>	LS-DYNA(V940)
PSHELL		SECTION_SHELL
PSOLID		SECTION_SOLID
PBAR		SECTION_BEAM(resultant)
PBEAM		SECTION_BEAM(resultant)
PELAS		SECTION_DISCRETE
PROD		SECTION_BEAM(truss)
PTUBE		SECTION_BEAM(integrated)

C. MATERIAL PROPERTY:

NASTRAN	===>	LS-DYNA(V940)
MAT1		1 -MAT_ELASTIC

D. BOUNDARY AND LOAD CONDITIONS:

NASTRAN	===>	LS-DYNA(V940)
SPC		BOUNDARY_SPC_SET
NODE FORCE		LOAD_NODE_SET
SHELL PRESSURE		LOAD_SHELL_SET
INITIAL VELOCITY		INITIAL_VELOCITY

E. OTHERS:

Convert NODE SET, ELEMENT SET.

NASTRAN	===>	LS-DYNA(V940)
LOAD CURVE		DEFINE_CURVE
LOCAL COORDINATE		DEFINE_COORDINATE_SYSTEM

Appendix G
Converting LS-DYNA to NASTRAN

Element, property and condition equivalancies are displayed below for LS-DYNA to NASTRAN conversion.

A. ELEMENT:

Delete SEATBELT elements, JOINT elements.

LS-DYNA(V940) ===>	NASTRAN
BEAM	CBAR, CBEAM, CROD, CTUBE (according to it's element property)
CONSTRAINED_NODAL_RIGID_BODY	RBE2
T-SHELL	SOLID
SHELL	SHELL
SOLID	SOLID
DISCRETE	CELAS1 (create local coordinate)
MASS	CONM2
CONSTRAINED_SPOTWELD	RBAR
INERTIA	CONM2
CONSTRAINED_NODE_SET	RBE2

B. ELEMENT PROPERTY:

Delete SEATBELT section.
Put spring stiffness into element properties.

LS-DYNA(V940) ===>	NASTRAN
SECTION_BEAM(integrated)	PBAR, PTUBE (according to it's ELFORM and CST)
SECTION_BEAM(resultant)	PBAR
SECTION_BEAM(discrete)	PBAR
SECTION_BEAM(truss)	PROD
SECTION_BEAM(spotweld)	PBAR
SECTION_DISCRETE	PELAS
SECTION_SHELL	PSHELL

SECTION_SOLID	PSOLID
SECTION_SOLID_ALE	PSOLID
SECTION_TSHELL	PSOLID

C. MATERIAL PROPERTY:

All materials convert into MAT1.

D. BOUNDARY AND LOAD CONDITION:

LS-DYNA(V940)	===>	NASTRAN
BOUNDARY_SPC_NODE		SPC
BOUNDARY_SPC_SET		SPC
INITIAL_VELOCITY		INITIAL VELOCITY
INITIAL_VELOCITY_NODE		INITIAL VELOCITY
INITIAL_VELOCITY_GENERATION		INITIAL VELOCITY
LOAD_NODE_POINT		NODE FORCE
LOAD_NODE_SET		NODE FORCE
LOAD_SHELL_ELEMENT		SHELL PRESSURE
LOAD_SHELL_SET		SHELL PRESSURE

E. OTHERS:

Convert NODE SET, ELEMENT SET.
Delete all others which can't be converted.

LS-DYNA(V940)	===>	NASTRAN
DEFINE_COORDINATE_NODES		LOCAL COORDINATE
DEFINE_COORDINATE_SYSTEM		LOCAL COORDINATE
DEFINE_COORDINATE_VECTOR		LOCAL COORDINATE
DEFINE_CURVE		LOAD CURVE

Appendix H
Converting RADIOSS to NASTRAN

Element, property and condition equivalancies are displayed below for RADIOSS to NASTRAN conversion.

A. ELEMENT:

RADIOSS(V3.1)	===>	NASTRAN
BEAM		CBAR
TRUSS		CROD
SOLID		SOLID
SHELL		SHELL
SPRING		CELAS1 (lost the third node.)
MASS		CONM2
JOINT		RBE2
RIGID BODY		RBE2
SPOTWELD		RBAR

B. ELEMENT PROPERTY:

Delete VOID, RIVET, AIRBAG property.

RADIOSS(V3.1)	===>	NASTRAN
1 -SHELL		PSHELL
2 -TRUSS		PROD
3 -BEAM		PBAR
4 -SPRING		PELAS
6 -ORTHOTROPIC SOLID		PSOLID
8 -GENERAL SPRING		PELAS
9 -ORTHOTROPIC SHELL		PSHELL
10-COMP. SHELL		PSHELL
11-COMP. SHELL		PSHELL
12-3-NODES SPRING		PELAS
13-BEAM TYPE SPRING		PELAS
14-GENERAL SOLID		PSOLID

C. MATERIAL PROPERTY:

All materials convert into MAT1.

D. BOUNDARY AND LOAD CONDITION:

Delete PRESSURE LOAD.

RADIOSS(V3.1)	===>	NASTRAN
SPC		SPC
CONCENTRATED LOAD		NODE FORCE
INITIAL VELOCITY		INITIAL VELOCITY

E. OTHERS:

Delete NODE SET, SHELL SET, BEAM SET, SOLID SET, SPRING SET, TRUSS SET.
Delete all others which can't be converted.

RADIOSS(V3.1)	===>	NASTRAN
FUNCTION		LOAD CURVE
SKEW(moving)		LOCAL COORDINATE
SKEW(fixed)		LOCAL COORDINATE

Appendix I
Converting NASTRAN to RADIOSS

Element, property and condition equivalancies are displayed below for NASTRAN to RADIOSS conversion.

A. ELEMENT:

Delete RBE3 element, CGAP element.

NASTRAN	===>	RADIOSS(V3.1)
CBAR		BEAM
RBE2		RIGID BODY
SHELL		SHELL
SOLID		SOLID
CELAS2		SPRING
CELAS1		SPRING
RBAR		SPOTWELD
CONM2		MASS
CBEAM		BEAM
CROD		TRUSS
CTUBE		BEAM
ISOPARAMETRIC SHELL		SHELL

B. ELEMENT PROPERTY:

Delete PGAP property.

NASTRAN	===>	RADIOSS(V3.1)
PSHELL		1-SHELL
PSOLID		14-GENERAL_SOLID
PBAR		3-BEAM
PBEAM		3-BEAM
PELAS		4-SPRING
PROD		2-TRUSS
PTUBE		3-BEAM

C. MATERIAL PROPERTY:

NASTRAN	===>	RADIOSS(V3.1)
MAT1		1 -ELASTIC

D. BOUNDARY AND LOAD CONDITIONS:

NASTRAN	===>	RADIOSS(V3.1)
SPC		SPC
NODE FORCE		CONCENTRATED LOAD
SHELL PRESSURE		PRESSURE LOAD
INITIAL VELOCITY		INITIAL_VELOCITY

E. OTHERS:

Delete NODE SET, ELEMENT SET.

NASTRAN	===>	RADIOSS(V3.1)
LOAD CURVE		FUNCTION
LOCAL COORDINATE		SKEW_MOV

Appendix J
Converting ADAMS to LS-DYNA

A typical ADAMS model is generated with markers and parts. The markers generally locate important hard points, bushing seats and mechanism connections. Parts ususally represent rigid body entities, which are connected by different joints, deformable beams, and discrete springs/ dampers.

VPG's ADAMS interface converts all markers to nodes, and all parts to the PART_INERTIA keyword card with mass-inertia property defined in the part LPRF. Bushings and springs/dampers are transferred as linear/nonlinear discrete elements. If it is a nonlinear discrete property, LS-DYNA uses the "load curve" function to define its characteristics.

The following subsections explain the process and capabilities of converting an ADAMS model to VPG.

A GENERAL DESCRIPTION OF ADAMS MODELING

Model Build:

- Define markers; a marker is a position with three-dimensional orientation.
- Define Parts with mass and inertia properties.
- Construct constrained joints, connecting beams and bushings.
- Apply proper input forces to the model.
- 95% of ADAMS models are built with rigid body parts with concern about the dynamic performance of those parts.

System Elements:

- System elements describe model subsystems that have dynamics of their own.
- System elements contain state variables, differential equations, linear state equations, transfer functions, user-defined subroutines and general state equations.

Data Elements:

- Array defines a list of input values, variables, or initial conditions.
- Curve defines a three-dimensional parametric curve.
- Spline defines discrete data for interpolation

Tire:

- Four tire types:
  *Fiala *UA *Smithers *Delft
- Inputs to calculate tire forces:
  *Mass *Slip Angle *Longitudinal slip ratio
  *Vertical stiffness *Rolling resistance * Radius
  *Friction coefficient with/without sliding
- Tire analysis outputs:
  *Contact patch *Forces at contact patch
  *Lateral slip angle, slip ratio, inclination angle and normal deflection
  *Friction ellipse properties

Validate

- Study the degree-of-freedom of the whole model.
- Import test data to compare with ADAMS/View Data

Refine

- Define design variables and parameterize the critical point locations.
- Calculate sensitivity of each design variable.
- Find the static equilibrium position

Optimize

- Adjust design variables to minimize or maximize a particular aspect of the model's performance.
- Ensure to improve the objective as much as possible without violating all constraints.
- Evaluate the model and reduce variations between runs.

Animate & Output

- Dynamic, Static, Assemble and Linearized simulations
- An assemble simulation attempts to correct any broken joints.

- Output predefined disp., vel., acc. and forces; user-defined measures and requests

CAPABILITY OF READING ADAMS TO VPG

Units:

- ADAMS: Force, Mass, Length, Time
- VPG/Pre-Processing: Five options
- VPG/Solver-DYNA: Three basic unit sets: M, Sec, Kg, N; MM, Sec, Ton, N; MM, Msec, Kg, KN

Graphics:

- ADAMS: Outline, Box, Circle, Arc, Cylinder
- VPG/Pre-Processing: Point, Line, Surface
- VPG/Solver-DYNA: N/A

Marker:

- ADAMS: Position with orientation
- VPG/Pre-Processing: Node with a local coordinate system
- VPG/Solver-DYNA: *NODE, and *CONSTRAINED_NODAL_RIGID_BODY contains unreferenced nodes.

Part:

- ADAMS: Mass, CG, Inertia Tensor, Initial transnational/rotational velocity
- VPG/Pre-Processing: Create a part with inertia properties defined in part local coordinate system and assigned a rigid material property
- VPG/Solver-DYNA: *PART_INERTIA with IRCS=1

Joints:

- ADAMS: Revolute, Cylindrical, Fixed, Hooke, Planar, Universal, Screw, Transnational, Rack-and-Pinion
- VPG/Pre-Processing: To meet Dyna joint initial position coincidence requirement, additional nodes are created along relative marker orientation
- VPG/Solver-DYNA: CONSTRAINED_JOINT_options

Bushing:

- ADAMS: Linear stiffness and damping
- VPG/Pre-Processing: 3 translational springs plus
  3 rotational springs plus
  3 translational dampers plus
  3 rotational dampers
- VPG/Solver-DYNA: 12 *ELEMENT_DISCRETE
  12 *SECTION_DISCRETE
  12 *MAT_options

Springdamper:

- ADAMS: A rotational or translational springdamper
- VPG/Pre-Processing: 1 translational spring plus
  1 rotational spring plus
  1 translational damper plus
  1 rotational damper
- VPG/Solver-DYNA: 4 *ELEMENT_DISCRETE
  4 *SECTION_DISCRETE
  4 *MAT_options

Beam:

- ADAMS: A massless Beam with a uniform cross section
- VPG/Pre-Processing: A beam element with flexible material and integration cross section
- VPG/Solver-DYNA: 1 *ELEMENT_BEAM
  1 *SECTION_BEAM
  1 *MAT_ELASTIC

Spline:

- ADAMS: Two-dimensional discrete data
- VPG/Pre-Processing: A load Curve
- VPG/Solver-DYNA: *DEFINE_CURVE

CONVERTING AN ADAMS SUSPENSION TO VPG

In the following example, an ADAMS suspension is converted to VPG. The vehicle front suspension is a short-long arm mechanism. Almost all beam parts are rigid except the stabilizer bar, which connects to the lower control arm with a vertical link. Cylindrical joints exist between the knuckles and the wheel half shaft to constrain revolution motion.

The rear suspension, using leaf springs, is called a Hotchkiss mechanism. This leaf spring model is a basic kinematic model combined of several pivoting rigid beams. The vertical stiffness of the leaf springs is taken into account through a spring element connecting the upper pivoting beams to the lower ones. A torsional leaf spring is introduced to constrain a rotational degree of freedom between these lower pivoting beams.

An FEA tire model is constructed using the VPG software and applied to this example model. Basically, the tire model is a combination of plies, treads, sidewall and chafer, which are assigned different material properties. The tire also includes an airbag model to account for internal air pressure.

Conversion:

(1) The units conversion statement is read, and the user is offered four units (force, length, mass and time) to consider in correspondence with VPG database units.

(2) ADAMS graphic elements (outline, box, circle, arc, and cylinder) are converted to VPG geometric items.

(3) The ADAMS' part card is converted to the LS-DYNA card, *PART_INERTIA, with mass-inertia property defined in the part LPRF.

(4) All markers are transferred to the global coordinate system *NODE in LS-DYNA. The option of a local coordinate system is provided to refer to the original marker direction if necessary.

(5) The ADAMS beam card is changed to the LS-DYNA equivalent *ELEMENT_BEAM. The beam cross section property is stored in *SECTION_BEAM. A default *MAT_ELASTIC material property is assigned to the above beam element.

(6) All joints are translated as *CONSTRAINED_JOINT_option. Additional nodes are

generated in the marker LPRF coordinate system to meet the LS-DYNA joint initial position coincidence requirement.

(7) SPRINGDAMPER is defined as *ELEMENT_DISCRETE with stiffness and damping properties stored in *MAT_SPRING_ELASTIC or *MAT_DAMPING_VISCOUS.

(8) There are two ways to convert the ADAMS' bushing. In the first, a bushing is treated as a nonlinear beam element. The beam direction is given in its section property with a system coordinate defined. In the second, the bushing is split into three translational springs, three rotational springs, three translational dampers and three rotational dampers as needed. Load curves should be made for these nonlinear material properties.

(9) The spline statement is specified as *DEFINE_CURVE in LS-DYNA.

(10) Unreferenced markers are connected by rigid body beams or constrained nodal rigid bodies to their original parts. Otherwise, these markers become unreferenced nodes in VPG.

To offer a general idea of VPG efficiency, the converted model has 10371 nodes and 9526 elements, including rigid body representations of the body and chassis/suspension model and complete finite element tire models (4 tires-2150 elements per tire).

To simulate 1 second, it takes approximately 50 hours CPU time on a UNIX based workstation. CPU time distribution is:

Initialization	0.44
Element processing	85.66
Binary databases	0.04
ASCII database	0.07
Contact algorithm	12.38
Contact entities	0.00
Rigid bodies	1.42

Appendix K
VPG Menu

The VPG menu lists the following programs and translators:

VPG Version 2.0
VPG Version 2.0 Large
VPG Server Communications
PDGS, CGS & CHRYSLER vendor file to VPG line data
Beam Section Calculation program
DYNA3D post file to VPG history file
VPG line data to PDGS standard tape format
VPG line data to IGES format
VPG line data to DXF format
Transfer NASTRAN punch from local to global
Pack NASTRAN punch file
Re-arrange NASTRAN punch file from S. E. ANALYSIS
Re-arrange NASTRAN punch file from Frequency Response Analysis
Re-arrange NASTRAN punch file from Transient Analysis
PATRAN Neutral File to VPG Model Binary File
IDEAS Universal File to VPG Model Binary File
VPG Database File to PATRAN Neutral File
VPG Database File to IDEAS Universal File

These programs and translators are described in the following pages.

VPG

VPG supports 250,000 elements.

VPG LARGE

VPG large supports 500,000 elements.

VPG SERVER COMMUNICATIONS

This program contains utilities pertaining to VPG licenses in use on a server. For more information regarding this program, consult the VPG installation guide.

PDGS, CGS & CHRYSLER VENDOR FILE TO VPG LINE DATA

This program converts and filters the line data from PDGS, CGS, or Chrysler Vendor File format into VPG Neutral Line Data.

1. VPG prompts:

```
>   INPUT LINE DATA TYPE: (0,1,2,3,4)
?   0.  QUIT
    1.  PDGS
    2.  CGS
    3.  CHRYSLER VENDOR FILE
    4.  VPG LINE DATA
```

   - If the user selects the **CGS** format above, VPG prompts:

```
>   ENTER CGS DATA FORMAT:
?   0 = QUIT
    1 = INCA FORMAT
    2 = DES  FORMAT
```

   - An error will occur after step 7 if an incorrect data type is specified.

Continued on next page.

2. VPG prompts:
   > ENTER THE [line data type**] LINE DATA FILE NAME,** or **"STOP" TO RETURN TO THE MENU:**

3. VPG prompts:
   > FILTER OUT UNNECESSARY POINTS? (Y/N)
   - YES decreases file size and continues with step 5.
   - NO jumps to step 6.

4. VPG prompts for filter parameters:
   > DEFAULT PARAMETERS FOR FILTERING POINTS ARE:
   > DISTANCE < 5 PERCENT OF THE MODEL SIZE
   ANGLE BETWEEN LINE SEGMENTS < 11.25 DEGREE
   DO YOU WANT TO CHANGE THESE CRITERIA? (Y/N)
      - YES prompts:
        > ENTER PERCENT OF MODEL SIZE BETWEEN POINTS
        > ENTER ANGLE BETWEEN LINE SEGMENTS IN DEGREE
      - NO keeps the default values.

5. VPG prompts:
   > ENTER THE NETURAL LINE DATA FILE NAME OR "STOP"
      - If the file name exists, VPG prompts the user to overwrite it or to select a new name.

6. VPG displays:
   > START READING [LINE DATA TYPE**] LINE DATA**
   + XXXX RECORDS READ
   FILTERING LINE DATA (SEE STEP 4)
   XXXXX POINTS REDUCED TO XXXXX POINTS
   NEUTRAL LINE DATA FILE CREATED ===>>> FILENAME.LIN
   CONVERSION AND FILTER COMPLETE
   NORMAL TERMINATION.
   ..APPLICATION DONE..

BEAM SECTION CALCULATION PROGRAM

This program calculates the thin wall cross-section properties of beams. The calculated properties include cross-sectional area, polar moment of inertia, moment of inertia about two local axes, torsional constant, centroid location, and principle angle. Cross sections may be open or closed. The input data may be entered from the user terminal or from an external file. The calculated properties may be output to the user terminal as well as to an external file. The input data is described below.

notes:	
ISECT:	Section number (integer)
TYPE:	Section type (open or closed)
IPTN:	Point number (integer). Points must be in ascending order. Enter IPT = 0 (zero) to end a section.
YN, ZN:	Y and Z coordinate of point N (real numbers). Any coordinate system may be used for Y and Z. The location of points should be at the centerline of the thickness of the segment. Segment length must not be zero.
TN:	Thickness of the segment between point N and N+1 (real number). Value must be positive.

1. VPG prompts:
   > Input data from an external file? (y/n)
   - If yes:
     > Enter Beam Section Data File name or STOP
2. VPG prompts:
   > Output properties to external file?(y/n)
   - If yes:
     > Enter output file name
3. VPG prompts:
   > Enter section number & type (open/close)
   > For example: 10,open
4. VPG prompts:
   > Enter ipt, y, z, t
   > End section input with ipt = 0

Continued on next page.

5. VPG displays:

```
>   Processing Beam Section    XX
>   Hit [CR] key to continue
OPEN
SECTION
XX
AREA
X.XXXXE+XX
IYC
X.XXXXE+XX
IZC
X.XXXXE+XX
IYZ
X.XXXXE+XX
J
X.XXXXE+XX
IYP
X.XXXXE+XX
IZP
X.XXXXE+XX
YBAR
X.XXXXE+XX
ZBAR
X.XXXXE+XX
ANGLE
X.XXXXE+XX
```

6. VPG prompts:

```
>   DO YOU WANT TO TERMINATE? (y/n)
```

- If NO, VPG jumps to step 3.
- If YES, the program terminates.
- The output file created by the beam section program contains the following information:

```
BEAM CROSS - SECTION DATA
BEAM CROSS SECTION NO. XX **(open or closed) SECTION**
XX SEGMENTS IY              IZ                IT
XX             X.XXXXE+XX X.XXXXE+XX        X.XXXXE+XX
```

Continues on following page.

```
XX              X.XXXXE+XX  X.XXXXE+XX         X.XXXXE+XX
XX              X.XXXXE+XX  X.XXXXE+XX         X.XXXXE+XX
XX              X.XXXXE+XX  X.XXXXE+XX         X.XXXXE+XX
CALCULATED PROPERTIES
AREA        IY(USER) IZ(USER)    IYZ(USER)   J
X.XXXXXE+XXX.XXXXXE+XX  X.XXXXXE+XX  X.XXXXXE+XXX
    X.XXXXXE+XX
IYP         IZP          YBAR         ZBAR         ANGLE
X.XXXXXE+XXX.XXXXXE+XX  X.XXXXXE+XX  X.XXXXXE+XX
    X.XXXXXE+XX
```

DYNA3D POST FILE TO VPG HISTORY FILE

This program translates DYNA-3D state files into NASTRAN format such that VPG can plot deformed shapes and stress-contours. This program can either generate deformed geometry plots along with stress and strain at any or all time increments or it can process displacement and stress/strain animation. Input to the program is DYNA-3D state output file called D3PLOT for UNIX.

The program generates deformed geometry files DG____NN, where NN = 00,01,...,200 and the blanks are replaced by first four characters of the input file. Only one stress output file ____.str and one strain output file ____.stn are created in which the blanks are replaced by the first 4 characters of the input file with subcases 1,2,3,.,200.

DG____01 contains the undeformed geometry, while ____.ani contains the displacement & stress/strain animation data. D3PL.GLVB contains the global variable information. It is advised to read DG____01 as a NASTRAN file to create initial geometry in a new database. This avoids any errors due to internal renumbering by DYNA.

1. VPG prompts:
   > ENTER THE ROOT PLOT FILE NAME OR QUIT
   (DEFAULT = D3PLOT)
   - VPG displays a list of all plot files found.

Continues on following page.

2. VPG prompts:

```
>   SELECT POST-PROCESSING OPTION
?   QUIT
    ANIMATION (DEFAULT)
    INDIVIDUAL STATES
```

   - If animation is selected, VPG jumps to step 5.

3. VPG prompts:

```
>   INDIVIDUAL STATES WILL BE PROCESSED
>   ENTER STRESS/STRAIN FILE FORMAT
?   QUIT
    STANDARD PUNCH
    PACKED PUNCH (DEFAULT)
```

   - VPG jumps to step 7.

4. VPG prompts:

```
>   ANIMATION WILL BE PROCESSED
>   SELECT ONE OF THE FOLL. OPTIONS FOR ANIM.
?   QUIT
    DISPLACEMENT ONLY (DEFAULT)
    MENU TO SELECT STRESSES, STRAINS & ENERGY
```

   - If displacement only is selected, VPG jumps to step 7.

5. VPG prompts:

```
>   DISPLACEMENT AND STRESS WILL BE PROCESSED
>   SELECT UP TO SIX OF THE FOLLOWING OPTIONS FOR
    ANIMATION
(MID-SURFACE)      (BOTTOM-SURFACE)    (TOP-SURFACE)
_______-     ________ _______-
1.sigma-xx         8.sigma-xx          15.sigma-xx
2.sigma-yy         9.sigma-yy          16.sigma-yy
3.sigma-zz         10.sigma-zz         17.sigma-zz
4.sigma-xy         11.sigma-xy         18.sigma-xy
5.sigma-yz         12.sigma-yz         19.sigma-yz
6.sigma-zx         13.sigma-zx         20.sigma-zx
7.plastic strain   14.plastic strain   21.plastic
    strain
```

Continues on following page.

```
22.Thickness
23.Internal energy
24.Max. Vonmises stresses
25.Normal Strain
```

- The options must be separated by commas or spaces.
- VPG displays the options selected:

```
>   XX  XX    XX    0     0     0
```

6. VPG displays:

```
>   STATE & TIME INFORMATION OBTAINED FROM THE DATA-
    BASE
    TOTAL NUMBER OF STATES FOUND          =         XX
    TIME INTERVAL BETWEEN EACH STATE DUMP =   XXX.XXXX
    PLOT FILE=d3plot STATE NO.1 AT TIME =0.0000000E+00
    PLOT FILE=d3plot STATE NO.2 AT TIME =X.XXXXXXE+XX
    PLOT FILE=d3plot01 STATE NO.3 AT TIME =
    X.XXXXXXE+XX
```

7. VPG prompts:

```
>   SELECT STATE POST PROCESSING OPTION
?   QUIT
    ALL
    SELECTIVE (DEFAULT)
```

- If all is selected, VPG jumps to step 12.

8. VPG prompts:

```
>   YOU WILL BE ABLE TO SELECT THE STATES YOU WOULD
    LIKE TO BE PROCESSED
>   TO SELECT STATES CHOOSE ONE OF THE FOLLOWING
?   QUIT
    FIXED INCREMENT (DEFAULT)
    RANDOM
```

- If random is selected, VPG jumps to step 11.

9. VPG prompts:

```
>   THE INCREMENT BETWEEN FILES WILL BE FIXED
>   ENTER STATE INCREMENT FOR POST-PROCESSING THIS
    OPTION WILL OUTPUT ALL THE STATES STARTING WITH
```

Continues on following page.

```
STATE 1 AND INCREMENTING THE NEXT OUTPUT BY THE
STATE INCREMENT.  TYPE 0 TO QUIT
```

- The selected states are displayed on screen:

```
>   TIMETAB=          XX
    TIMETAB=          XX
    TIMETAB=          XX
```

  - VPG jumps to step 12.

10. VPG prompts:

```
>   THE INCREMENT BETWEEN FILES WILL BE USER DEFINED
>   ENTER STATE NUMBERS FOR POST-PROCESSING.  STATE
    SHOULD BE ONE INTEGER VALUE IN EACH LINE.  TYPE
    "END" TO FINISH STATE TABLE OR STOP TO QUIT
```

11. VPG displays:

```
>   Title from result file
    CODE IDENTIFICATION         ( ICODE )=        XXXX
    PROBLEM DIMENSION           (  NDIM )=        XXXX
    # OF NODES                  ( NUMNP )=        XXXX
    # OF 8 NODE ELEMENTS        (  NEL8 )=        XXXX
    # OF 8 NODE SHELL ELEMENTS  (  NELT )=        XXXX
    # OF 2 NODE ELEMENTS        (  NEL2 )=        XXXX
    # OF 4 NODE ELEMENTS        (  NEL4 )=        XXXX
    # OF GLOBAL VARIABLE        ( NGLBV )=        XXXX
    # OF VARIABLE 3D-ELM        (  NV3D )=        XXXX
    # OF VARIABLE 3D-SHELL ELM. ( NV3DT )=        XXXX
    # OF VARIABLE 2D-ELM        (  NV2D )=        XXXX
    # OF VARIABLE 1D-ELM        (  NV1D )=        XXXX
    # OF MATERIAL 3D-ELM        ( NUMAT8)=        XXXX
    # OF MATERIAL 3D-SHELL ELM. (NUMMATT)=        XXXX
    # OF MATERIAL 2D-ELM        ( NUMAT4)=        XXXX
    # OF MATERIAL 1D-ELM        ( NUMAT2)=        XXXX
    # OF INTEG. POINTS FOR SHELL( MAXINT)=        XXXX
    # TEMPERATURE  FLAG         (   IT  )=        XXXX
    # GEOMETRY     FLAG         (   IU  )=        XXXX
    # VELOCITY     FLAG         (   IV  )=        XXXX
    # ACCELERATION FLAG         (   IA  )=        XXXX
```

Continues on following page.

```
# OF AD VARB FOR NEL8       ( NEIPH )=        XXXX
# OF AD VARB FOR NEL4       ( NEIPS )=        XXXX
ARBITRARY NUMBERING STORAGE ( NARBS )=        XXXX
READING GEOMETRIC DATA ...
READING STATE DATA ...
!!! PROCESSING STATE AT TIME=    X.XXXXXXXE+XX
PROCESSING FILE d3plot01
!!! PROCESSING STATE AT TIME=   X.XXXXXXXE+XX
  ***** PROCESSING COMPLETED *****
Normal termination.
..Application done..
```

VPG LINE DATA TO PDGS STANDARD TAPE FORMAT

This translator converts VPG line data into PDGS format data.

1. VPG prompts:
   > ENTER THE VPG LINE DATA FILE NAME
   - VPG displays:
     VPG LINE DATA FILE NAME ====>> filename.lin

2. VPG prompts:
   > ENTER THE OUTPUT PDGS FILE NAME
   - If the file name exists, VPG prompts to overwrite it or to select a new file name.

3. VPG prompts:
   > ENTER TITLE FOR PDGS FILE (UP TO 24 CHARACTERS)
   - This option creates a title within the PDGS file.

4. VPG displays:
   > PDGS FILE HAS BEEN WRITTEN
   ..Application done..

VPG LINE DATA TO IGES FORMAT

This translator converts VPG line data into IGES line data format.

1. VPG prompts:
   > ENTER THE INPUT VPG LINE DATA FILE NAME
   - VPG displays:
     VPG LINE DATA FILE NAME ====>> filename.lin
   - The default output filename uses the same filename with the extension .igs.
   - If the file name exists, VPG prompts to overwrite file or select new file name.
   - VPG displays:
     OUTPUT IGES FILE NAME ========>> filename.igs
2. VPG prompts:
   > ENTER TITLE FOR IGES FILE (UP TO 24 CHARACTERS)
   - Creates title within IGES file.
3. VPG prompts:
   > ENTER THE RATIO OF MODEL SPACE TO REAL WORLD SPACE (DEFAULT 1.0)
4. VPG prompts:
   > ENTER THE MEASURING UNIT USED IN THE FILE:
   1. INCHES
   2. MILLIMETERS
   3. FEET
   4. METERS
   5. CENTIMETERS
   6. OTHER
5. VPG prompts:
   > ENTER NAME OF AUTHOR (UP TO 24 CHARACTERS)
6. VPG prompts:
   > ENTER AUTHOR'S ORGANIZATION (UP TO 50 CHARACTERS)
7. VPG displays:
   > PROCESSING VPG DATA
   XXX VPG RECORDS READ
   Successfully completed processing
   The VPG data has been translated to the following IGES data :
   TYPE XXX : XXX RECORDS
   ..Application done..

VPG LINE DATA TO DXF FORMAT

This translator converts VPG line data into DXF format.

1. VPG prompts:
   > ENTER THE VPG LINE DATA FILE NAME
   - VPG displays:
     VPG LINE DATA FILE NAME ====>> filename.lin

2. VPG prompts:
   > ENTER THE OUTPUT DXF FILE NAME
   - If the file name exists, VPG prompts to overwrite file or select new file name.
   - VPG displays:
     OUTPUT DXF FILE NAME ========>> filename.dxf

3. VPG displays:
   > DXF FILE HAS BEEN WRITTEN
   ..Application done..

TRANSFER NASTRAN PUNCH FROM LOCAL TO GLOBAL

This program transforms displacement vectors from a local coordinate system into a global coordinate system in a NASTRAN punch file. The program writes a new punch file for post processing. If the stress/strain results are included in the original punch file, they will be included in the new punch file. The corresponding NASTRAN bulk data file is required to provide the local coordinates data.

note: Only the displacements for the nodes in the bulk data file will be included in the new punch file. The list file **trans.msg** will contain messages related to the coordinate transformation.

1. VPG prompts:
   > ENTER THE NASTRAN PUNCH FILE NAME OR STOP
   - VPG displays:
     NASTRAN PUNCH FILE ========>> filename.pch

2. VPG prompts:
   > IS THE NASTRAN PUNCH FILE PACKED? (Y/N)

Continues on following page.

3. VPG prompts:
   ```
   >   ENTER THE NASTRAN BULK DATA FILE NAME OR STOP TO
       EXIT
   ```
   - VPG displays:
     ```
     NASTRAN BULK DATA FILE ====>> filename.nas
     ```

4. VPG prompts:
   ```
   >   ENTER THE TRANSFORMED PUNCH FILE NAME OR STOP TO
       EXIT
   ```
   - VPG displays:
     ```
     TRANSFORMED PUNCH FILE ====>> filename.pch
     ```

5. VPG displays:
   ```
   >   Reading and sorting bulk data
       Reading coordinate frames
   ```

PACK NASTRAN PUNCH FILE

This program packs the nodal displacement, mode shapes, and element stresses from a NASTRAN punch file, and writes the packed output onto another file.

1. VPG prompts:
   ```
   >   ENTER NASTRAN PUNCH FILE NAME OR STOP
   ```

2. VPG prompts:
   ```
   >   ENTER PACKED STRESS FILE NAME OR STOP
   ```

3. VPG displays:
   ```
   >   PROCESSING SUBCASE X
       PROCESSING SUBCASE X
       NUMBER OF DISPLACEMENT COMPACTED XXXX
       NUMBER OF PLATE ELEMENT STRESS COMPACTED XXXX
       ..Application done..
   ```

RE-ARRANGE NASTRAN PUNCH FILE FROM S. E. ANALYSIS

This option regroups elements from super-element collectors by subcase.

1. VPG prompts:
   ```
   >   ENTER NASTRAN PUNCH FILE NAME FROM SUPER ELEMENT
       RUN OR STOP
   ```

2. VPG prompts:
   ```
   >   ENTER RE-ARRANGED PUNCH FILE NAME OR STOP
   ```

3. VPG displays:
   ```
   >   PROCESSING DISPLACEMENT SUBCASE            XX
       PROCESSING STRESS/STRAIN SUBCASE       XX
       PROCESSING DISPLACEMENT SUBCASE        XX
       PROCESSING STRESS/STRAIN SUBCASE       XX
       PROCESSING DISPLACEMENT SUBCASE        XX
       PROCESSING STRESS/STRAIN SUBCASE       XX
       WRITING NEW NASTRAN PUNCH FILE
       ..Application done..
   ```

RE-ARANGE NASTRAN PUNCH FILE FROM FREQUENCY RESPONSE ANALYSIS

This program re-arranges displacement and stresses in a punch file generated from a NASTRAN frequency-response analysis.

1. VPG prompts:
   ```
   >   ENTER NASTRAN PUNCH FILE NAME FROM FREQUENCY RE-
       SPONSE OR STOP
   ```

2. VPG prompts:
   ```
   >   CREATE X-Y PLOT FILE INSTEAD OF NEW PUNCH FILE?
       (Y/N)
   ```

3. If Yes is selected,
   VPG prompts:
   ```
   >   ENTER NASTRAN TYPE
   ```

Continues on following page.

```
0 - ABORT
1 - MSC?NASTRAN
2 - CSA/NASTRAN
3 - UAI/NASTRAN
```

4. After the NASTRAN type is selected, VPG displays:

```
>   EXECUTE VPG PROGRAM WITH THE FOLLOWING
    TH - TIME HISTORY PLOT
    NG - NEW GRAPH
    FC - VPG CURVE LINE
    2  - COLUMN DIRECTED
    SELECT X-Y PLOT FILE NAME
    RD - READ DATABASE
    NG - NEW GRAPH
    FC - VPG CURVE FILES
    SELECT X-Y PLOT FILE NAME

… Application done …
```

5. If NO is selected after step 3,
   VPG prompts:

```
>   ENTER RE-ARRANGE PUNCH FILE NAME OR STOP
```

6. After the punch file name is entered,
   VPG prompts:

```
>   ENTER NASTRAN TYPE.
    0 - ABORT
    1 - MSC?NASTRAN
    2 - CSA/NASTRAN
    3 - UAI/NASTRAN
```

7. After the NASTRAN type is entered,
   VPG Prompts:

```
>   PROCESS DISPLACEMENT OR STRESS RESULT (D/S)
```

8. Displacement or stress results will be processed as indicated by the user.

REARRANGE NASTRAN PUNCH FILE FROM TRANSIENT ANALYSIS

This program re-arranges displacement and stresses in a punch file generated from NASTRAN transient analysis.

1. VPG prompts:
   > ENTER NASTRAN PUNCH FILE NAME FROM TRANSIENT ANALYSIS OR STOP TO EXIT

2. VPG prompts:
   > CREATE AN ANIMATION PUNCH FILE INSTEAD OF PUNCH FILE?(Y/N)

3. If Yes is selected,
   VPG prompts:
   > ENTER VPG HISTORY FILE NAME OR STOP TO EXIT

4. After a file name is entered:
   VPG prompts:
   > ENTER NASTRAN BULK DATA FILE NAME OR STOP TO EXIT

5. After a file name is entered,
   VPG prompts:
   > ENTER THE SCALE FACTOR FOR THE DISPLACEMENT

6. After the scale factor is entered,
   VPG displays:
   ```
   READING NODAL DATA
   xxxx GRIDS PROCESSED
   WRITING VPG HISTORY FILE
   … Application done …
   ```

7. If NO is entered after step 3
   VPG prompts:
   > ENTER RE-ARRANGED PUNCH FILE NAME OR STOP

8. VPG displays:
   ```
   WRITING NEW NASTRAN PUNCH FILE
   …  Application done …
   ```

PATRAN NEUTRAL FILE TO VPG MODEL BINARY FILE

This translator converts PATRAN neutral files into VPG database files.

1. VPG prompts:
   ```
   >   PLEASE ENTER PATRAN NEUTRAL FILE NAME
       THE NETRUAL FILE TYPE MUST BE TEXT FORMAT
       (MAXINUM 20 CHARACTERS)    stop to QUIT
   ```

2. VPG prompts:
   ```
   >   PLEASE ENTER VPG DATABASE FILE NAME
       (MAXINUM 20 CHARACTERS)   stop to QUIT
   ```
   - If the file name exists, VPG prompts to overwrite file or select new file name.

3. VPG displays:
   ```
   >   TOTAL RECORD NUMBER IS        XXXX
       ..Application done..
   ```

IDEAS UNIVERSAL FILE TO VPG MODEL BINARY FILE

This translator converts I-DEAS Universal Files into VPG Database files.

1. VPG prompts:
   ```
   >   ENTER I-DEAS UNIVERSAL FILE NAME
       (MAXINUM 24 CHARACTERS OR STOP TO QUIT)
   ```

2. VPG prompts:
   ```
   >   ENTER VPG DATABASE FILE NAME
       (MAXINUM 24 CHARACTERS OR STOP TO QUIT)
   ```
   - If the file name exists, VPG prompts to overwrite file or select new file name.

3. VPG displays:

```
>   ***********************************************
    *          SUMMARY OF THE MODEL                *
    *          ----------———————————              *
    *    TOTAL NUMBER OF PART       :        XXX   *
    *    TOTAL NUMBER OF NODE       :        XXX   *
    *    TOTAL NUMBER OF ELEMENT    :        XXX   *
    *                                              *
    *  CHECK FILE ideasfmb.msg FOR ERROR MESSAGES  *
    *                                              *
    ***********************************************
    ..Application done..
```

VPG DATABASE FILE TO PATRAN NEUTRAL FILE

This translator converts VPG database files into PATRAN Neutral Files.

1. VPG prompts:
   ```
   >   PLEASE ENTER VPG DATABASE  FILE NAME
       (MAXINUM 20 CHARACTERS)    stop  TO QUIT
   ```

2. VPG prompts:
   ```
   >   PLEASE ENTER PATRAN NEUTRAL FILE NAME
       (MAXINUM 20 CHARACTERS)   stop  TO QUIT
   ```
   - If the file name exists, VPG prompts to overwrite file or select new file name.

3. VPG displays:
   ```
   >   ..Application done..
   ```

VPG DATABASE FILE TO IDEAS UNIVERSAL FILE

This translator converts VPG database files into I-DEAS Universal Files.

1. **VPG Database File to IDEAS Universal File**

Continued on next page.

2. VPG prompts:

   > ENTER VPG DATABASE FILE NAME
   (MAXINUM 24 CHARACTERS OR “STOP” TO QUIT

3. VPG prompts:

   > ENTER I-DEAS UNIVERSAL FILE NAME
   (MAXINUM 24 CHARACTERS OR “STOP” TO QUIT

   - If the file name exists, VPG prompts to overwrite file or select new file name.

4. VPG displays:

   > ** PLEASE CHECK FILE : fmbunv.msg **
   ..Application done..

   - File contains any error messages.

Function Index

Symbols

12 LINE SOLID MESH 121
2 LINE MESH 116
2 LINE SURFACE 70
3 LINE MESH 117
3 LINE SURFACE 70
3 POINT ARC 52
4 LINE MESH 118
4 LINE SURFACE 71
6 LINE SOLID MESH 119
8 LINE SOLID MESH 120
9 LINE SOLID MESH 121

A

ACTIVE WINDOW 357
ACTIVE WINDOW - ACTIVE WINDOW IS OFF 358
ACTIVE WINDOW - DIAGONAL NODE/POINTS 357
ACTIVE WINDOW - DRAG WINDOW 358
ACTIVE WINDOW - KEY IN XYZ RANGE 358
ACTIVE WINDOW - SCALE ACTIVE WINDOW 358
ACTWIN (ACTIVE WINDOW) 361
ADD CURVE TO GRAPH 308
ADD ELEMENTS TO PART 366
ADD LINES TO PART 366
ADD POINTS 50
ADD SURFACES TO PART 367
AIRBAGS 238
ALE 238
ANALYSIS PROGRAM 378
ANGLE 384
ANIMATE CONTOUR 296
ANIMATE DEFORMATION 296
ANNOTATE 154
ARC TANGENT 2 LINES 52
ASCII 241
ASPECT RATIO 141

ASSIGN MATERIAL 212, 218
ASSIGN PROPERTY 226
ASSIGN RESIDUAL 256
ASSIGN SUPERELEMENT I.D. 256
ATTRIBUTE TABLE ON/OFF 122
AUTO NORMAL 140
AUTO ORIENT SOLID 140
AUTO SEGMENT END 378

B

BINARY 242
BOUNDARY CARDS 174
BOUNDARY CARDS - CONVECTION 174
BOUNDARY CARDS - CYCLIC 175
BOUNDARY CARDS - FLUX 175
BOUNDARY CARDS - NON-REFLECTING 175
BOUNDARY CARDS - NON-REFLECTING2D 175
BOUNDARY CARDS - PRESCRIBED MOTION 176
BOUNDARY CARDS - PRESSURE OUTFLOW 176
BOUNDARY CARDS - RADIATION 176
BOUNDARY CARDS - SLIDING PLANE 176
BOUNDARY CARDS - SPC 177
BOUNDARY CARDS - SYMMETRY FAILURE 178
BOUNDARY DISPLAY 142
BOUNDARY LINE 69
BOX 247
BUTTERWORTH FILTER 312

C

CHANGE COLOR 226
CHANGE COLOR - MATERIAL 212, 218
CHANGE COLOR MAP 379
CHANGE COLORS 348
CHANGE COLORS - CHANGE COLOR MAP 349
CHANGE COLORS - CHANGE PART COLORS 348
CHANGE COLORS - COLOR BAR 350
CHANGE COLORS - FILL COLOR ON/OFF 349
CHANGE COLORS - PART COLORS ON/OFF 349

CHANGE COLORS - RESET COLOR MAP 349
CHANGE ID 87, 125
CHANGE PART COLORS 369
CHANGE SUPERELEMENT I.D. 256
CHECK ANGLE 143
CHECK DUPLICATE ID 125
CHECK FREE SURFACE 151
CHECK JACOBIAN 144
CHECK NORMAL 145
CHECK OVERLAP 144
CHECK RIGID LINK 145
CHECK SIZE 146
CHECK SPRING 152
CHECK TAPER 148
CHECK WARPAGE 148
CLEANUP MEMORY 309
CLEAR 399
CLEAR SCREEN 314
COARSE ELEMENT 86
COINCIDENT CHECK 126
COMBINE 264
COMBINE LINES 54
COMPACT NODE ID 128
CONNECTING ELEMENTS 367
CONNECTING NODES 368
CONSTRAINT CARDS 178 - 183
CONTOUR FILL 297
CONTOUR LINES 297
CONTOUR MODE 300
CONTROL CARDS 239
CONTROL POINT MESH 378
COORDINATE SYSTEM 248, 384
COORDINATE SYSTEM - CREATE 385
COORDINATE SYSTEM - CURRENT 387
COORDINATE SYSTEM - DELETE 386
COORDINATE SYSTEM - DISPLAY ON/OFF 387
COORDINATE SYSTEM - MODIFY 386
COPY - LINE 53
COPY - SURFACE 69
COPY - ELEMENT 87

COPY - NODE 127
COPY - MATERIAL PROPERTY (LS-DYNA) 213
COPY - MATERIAL PROPERTY (NASTRAN) 219
COPY - CONTACT INTERFACE 234
COPY - ELEMENT PROPERTIES (NASTRAN) 226
COPY - ELEMENT PROPERTIES (LS-DYNA) 227
COPY TIRE MODEL 282
CORNER BIAS MESH 379
CREATE - ELEMENT 90
CREATE - SET MENU 154
CREATE - MATERIAL PROPERTY (LS-DYNA) 213
CREATE - MATERIAL PROPERTY (NASTRAN) 219
CREATE - CONTACT INTERFACE 234
CREATE - SUSPENSION 270
CREATE - PART CONTROL 369
CREATE - ELEMENT PROPERTIES (LS-DYNA) 228
CREATE - ELEMENT PORPERTIES (NASTRAN) 227
CREATE (NODE) 128
CREATE ROAD SURFACE 262
CREATE S-LINES 70
CREATE SET (ELEMENTS) 164
CREATE SET (NODES) 160
CREATE SET (PARTS) 169
CREATE SUBCASE 192
CROSS PLOT 313
CROSS SECTION 243
CURRENT PART 399
CURRENT 155, 370
CURRENT SET (ELEMENTS) 166
CURRENT SET (NODES) 161
CURRENT SET (PARTS) 169
CURSOR ZOOM 354
CURVE 248
CURVE OPERATIONS 309

D

DAMPING CARDS 239
DATABASE (STAT)ISTICS 360
DATABASE 240
DATABASE - ASCII 241

DATABASE - CROSS SECTION 243
DATABASE - HISTORY 244
DATABASE - SPRING FORWARD 245
DATABASE - TRACER 246
DATABASE - EXTENT 244
DATABASE - BINARY 242
DATABASE - NODAL FORCE GROUP 245
DATABASE - SUPERPLASTIC FORMING 246
DEFINE CARDS 246
DEFINE CARDS - BOX 247
DEFINE CARDS - COORDINATE SYSTEM 248
DEFINE CARDS - VECTOR 250
DEFINE CARDS - CURVE 248
DEFINE CARDS - SPRING ORIENTATION 249
DEFINE PROPERTIES 214, 220
DEFINE PROPERTIES (NASTRAN) 230
DEFINE SECTION (LS-DYNA) 230
DEFINE TITLE 353, 388
DEFORMABLE TO RIGID 251
DEFORMED SHAPE 298
DELETE - LINE 56
DELETE - ELEMENT 104
DELETE - SET MENU (LS-DYNA) 155
DELETE - CONTACT INTERFACE 234
DELETE - ROAD SURFACE 262
DELETE - SUSPENSION 277
DELETE - PART 370
DELETE - ELEMENT PROPERTIES (LS-DYNA) 229
DELETE - ELEMENT PROPERTIES (NASTRAN) 229
DELETE CURVES 314
DELETE MATERIALS 213, 220
DELETE SET (ELEMENTS) 166
DELETE SET (NODES) 161
DELETE SET (PARTS) 170
DELETE SUBCASE 192
DELETE SURFACES 70
DELETE TIRE MODEL 282
DELETE UNREF. NODES 129
DEPTH CUEING 356

DISPLAY OPTIONS 300, 301, 304
DISTANCE 129, 388
DRAG MESH 104
DRAG MESH1 - ONE LINE DRAG 105
DRAG MESH2 - NORMAL DRAG 105
DRAG MESH3 OR 4 LINE DRAG 106
DRAW ARROW 387
DYNAMIC SCREEN X ROTATION 361
DYNAMIC SCREEN Y ROTATION 362
DYNAMIC SCREEN Z ROTATION 362
DYNAMIC VIRTUAL X ROTATION 362
DYNAMIC VIRTUAL Y ROTATION 362
DYNAMIC VIRTUAL Z ROTATION 363

E

EDGE BIAS MESH 379
ELEMENT SIZE 379
ELEMENT STRESS 298
ELEMENT TYPE 107
ELEMENT TYPE DISPLAY 380
EQUATION OF STATE 251
ERASE EMPTY SETS (NODE) 156
ERASE UNREF. SETS (NODE) 156
ERASE UNREFERENCED MATERIAL 214
ERASE UNREFERENCED SECTION 230
EXECUTE FATIGUE 293
EXTEND LINE 56
EXTENT 244
EXTERNAL 313
EXTRA NODES 179

F

FASTER SPEED 305
FATIGUE OPTION 292
FEATURE LINE DISPLAY 150
FILL 364
FILL COLOR 398
FILL COLOR ON/OFF 350
FILL HIDE PLOT 350

FILLHIDE 398
FIR FILTER 311
FLIP NORMALS 356
FREE ROTATION 380

G

GENERALIZED WELD 179
GENERATE ARCS 51
GENERATE LINES 56
GENERATE TIRE MODEL 280

H

HARDCOPY 315
HIDDEN SURFACE REMOVAL 356
HISTORY 244
HOURGLASS CARD 252

I

IDENTIFY 57
IDENTIFY ELEMENTS 107, 390
IDENTIFY NODE/POINT 130, 257
IDENTIFY NODES/POINTS 389
IDENTIFY SUPERELEMENT I.D. 257
INCLUDE 156
INCLUDE (ELEMENTS) 167
INCLUDE (PARTS) 170
INCLUDE NODES 162
INDIVIDUAL FRAME 305
INIT. COND. CARDS 184
INIT. COND. CARDS - DETONATION 184
INIT. COND. CARDS - MOMENTUM 185
INIT. COND. CARDS - STRESS 185
INIT. COND. CARDS - TEMPERATURE 185
INIT. COND. CARDS - VELOCITY 185
INIT. COND. CARDS - VOID 186
INITIAL VELOCITY 207
INITIAL VELOCITY - CLEAR INITIAL VELOCITY 208
INITIAL VELOCITY - DEFINE INITIAL VELOCITY 208
INITIAL VELOCITY - LIST INITIAL VELOCITY 208

INITIAL VELOCITY - REMOVE INITIAL VELOCITY 209
INITIAL VELOCITY - SHOW INITIAL VELOCITY 209
INITIAL VELOCITY - SHOW SPC 209
INTEGRATE 310
INTERFACE CARDS 252
ISO. SHELL FROM PLATE 108
ISOMETRIC 363

J

JOINT 102
JOINT STIFFNESS 180

K

KEEP PART 371

L

LABEL 108, 130
LAGRANGE IN SOLID 181
LINE MESH 110
LINEAR 181
LINES 398
LIST - SET 156
LIST - CONTACT CARDS 235
LIST - PART 372
LIST - ELEMENT PROPERTIES (LS-DYNA) 230
LIST - ELEMENT PROPERTIES (NASTRAN) 230
LIST CONTOUR VALUE 303
LIST DISPLACEMENT 302
LIST MATERIALS 215, 221
LOAD CARDS 186
LOAD CARDS - BEAM 187
LOAD CARDS - BODY 187
LOAD CARDS - BRODE 187
LOAD CARDS - DENSITY 187
LOAD CARDS - HEAT GENERATION 188
LOAD CARDS - NODE 188
LOAD CARDS - RIGID BODY 188
LOAD CARDS - SEGMENT PRESSURE 188
LOAD CARDS - SHELL PRESSURE 189

LOAD CARDS - SUB-SEA 189
LOAD CARDS - SUPER PLASTIC FORMING 189
LOAD CARDS - THERMAL 189
LOAD CURVE 391
LOAD CURVE - CREATE 391
LOAD CURVE - DELETE 392
LOAD CURVE - LIST 392
LOAD CURVE - MODIFY 392
LOAD CURVE - READ CURVE 393
LOAD CURVE - SHOW CURVE 394
LOADING 193
LOADING OPTIONS - CLEAR FORCE 196
LOADING OPTIONS - CLEAR PRESSURE 196
LOADING OPTIONS - CURRENT SET 195
LOADING OPTIONS - DEFINE FORCE 197
LOADING OPTIONS - DEFINE PRESSURE 197
LOADING OPTIONS - DELETE SET 196
LOADING OPTIONS - LIST FORCE 197
LOADING OPTIONS - LIST PRESSURE 198
LOADING OPTIONS - NEW SET 194
LOADING OPTIONS - REMOVE FORCE 199
LOADING OPTIONS - REMOVE PRESSURE 200
LOADING OPTIONS - SHOW FORCE 200
LOADING OPTIONS - SHOW PRESSURE 200
LOADING OPTIONS - SHOW SPC 201

M

MAGNIFY WINDOW 350
MAPPING (LCS) 88
MASS CALCULATION 371
MATERIAL COLOR ON/OFF 215
MATERIAL PART COLOR 221
MIRROR 58, 109
MODIFY - ELEMENT 110
MODIFY - CONTACT INTERFACE 235
MODIFY - PART 375
MODIFY SUBCASE 193
MOVE 277
MOVE LIGHT SOURCE 357

MOVE NODE TO PT/NODE 131
MOVE ROAD SURFACE 262

N

NODAL FORCE GROUP 245
NODAL RIGID BODY 100, 181
NODAL VALUE CURVE 306
NODE SET 182, 183
NODES BETWEEN PT/NODES 124
NORMAL 398

O

OFFSET 59
ON/OFF - MATERIAL (LS-DYNA) 215
ON/OFF - MATERIAL (NASTRAN) 222
ON/OFF - ELEMENT PROPERTY 231
ON/OFF - PART 372
OPTIONS 315
OPTIONS - AXIS OPTIONS 315
OPTIONS - CURVE OPTIONS 316
OPTIONS - GRAPH 317
OPTIONS - GRID OPTIONS 317
OPTIONS - LABEL OPTIONS 319
OPTIONS - LEGEND 320
ORIEN. 398

P

PAN 364
PART (control) 360
PART CONNECTION 132
PART ON/OFF 306
PART SUMMARY 373
PLOT MULTI GRAPHS 320, 321
PLOTEL ELEMENT 23
PRINT 364
PROJECT 60, 132
PROPERTY/PART COLOR 231

Q

QUERY 323

R

RANGE 324
READ ABAQUS FILE 40
READ BINARY FILE 40
READ DATABASE 324
READ DATABASE - NEW GRAPH - NODAL POINT DATA 326
READ DATABASE - NEW GRAPH - AIRBAG STATISTICS 332
READ DATABASE - NEW GRAPH - BEAM ELEMENT DATA 336
READ DATABASE - NEW GRAPH - BRICK ELEMENT DATA 335
READ DATABASE - NEW GRAPH - COLUMN DIRECTED 340
READ DATABASE - NEW GRAPH - CROSS SECTION FORCE 331
READ DATABASE - NEW GRAPH - DISCRETE ELEMENT 329
READ DATABASE - NEW GRAPH - GLOBAL DATA 327
READ DATABASE - NEW GRAPH - INTERFACE ENERGIES 333
READ DATABASE - NEW GRAPH - JOINT FORCE 334
READ DATABASE - NEW GRAPH - LIST DIRECTED 339
READ DATABASE - NEW GRAPH - MATERIAL ENERGIES 328
READ DATABASE - NEW GRAPH - NODAL FORCE 338
READ DATABASE - NEW GRAPH - RESULTANT INTERFACE FORCE 329
READ DATABASE - NEW GRAPH - RIGID BODY DATA 333
READ DATABASE - NEW GRAPH - RIGID WALL FORCE 330
READ DATABASE - NEW GRAPH - SHELL ELEMENT STRAIN 336
READ DATABASE - NEW GRAPH - SHELL ELEMENT STRESS 337
READ DATABASE - NEW GRAPH - TAURUS DATABASE 338
READ DATABASE - CURRENT GRAPH 340
READ DATABASE - NEW GRAPH 325
READ DATABASE - NO GRAPH 341
READ DATABASE - SELECT GRAPH 341
READ F-CRASH FILE 41
READ LINE DATA FILE 41
READ LS-DYNA FILE 41
READ MATERIALS 222
READ NASTRAN BULK 41
READ PAMCRASH FILE 42
READ RADIOSS FILE 42

READ RESULT FILE 298
REAR 363
RECALL VIEW 351
REDRAW 398
REMOVE 157
REMOVE (ELEMENTS) 168
REMOVE (PARTS) 171
REMOVE NODES 163
RENAME 222, 231
RENUMBER 111
RENUMBER ID 223
RENUMBER NODES 134
RENUMBER PRID (Property ID) 232
REPEAT 351
REROT (FREEHAND ROTATION) 363
RESPACE LINE 60
RESPACE UV LINES 71
RESTART 287
RESULTANT 313
REVERSE DIRECTION 61
REVERSE NORMAL 71
REVERSE NORMALS 112
REVERSE ROTATION 351
REVIEW LIST 157
RIGID BODIES 182
RUN ANALYSIS 285

S

SAE FILTER 312
SAVE 264
SAVE (database) 360
SAVE CURVES 342
SAVE MPEG 306
SAVE VIEW 351
SCALE 73, 135, 310
SCALE LINES 61
SCREEN AXES ROTATION 352
SEATBELT 101, 253
SECTION CUT 62, 150, 303

SECTION REVOLUTION 75
SELECT FROM LIBRARY 265
SELECT GRAPH 343
SELECT STEP/COMP. 305
SELECT STEP/COMP. (/VAR.) 303
SEPARATE PART(S) 373
SET DEFAULTS 342
SETUP 361
SHADE 398
SHADING OPTIONS 380
SHELL TO SOLID 182
SHOW - SURFACE 75
SHOW - INTERFACE 236
SHOW LINES 63, 395
SHOW ROAD SURFACE 263
SHOW SET 157
SHOW SET (ELEMENTS) 168
SHOW SET (NODES) 163
SHOW SET (PARTS) 171
SHOW STRAIN-STRESS 223
SHOW SUBCASE 193
SHRINK 398
SIDE 363
SLOWER SPEED 306
SMOOTH 310
SPC 201
SPC OPTIONS - CLEAR SPC 204
SPC OPTIONS - NEW SET 202
SPC OPTIONS - CURRENT SET 203
SPC OPTIONS - DEFINE DOF 204
SPC OPTIONS - DELETE SET 204
SPC OPTIONS - LIST SPC 206
SPC OPTIONS - REMOVE SPC 206
SPC OPTIONS - SHOW FORCE 207
SPC OPTIONS - SHOW SPC 207
SPLINE CURVE 55
SPLIT - SURFACE 74

SPLIT - ELEMENT 112
SPLIT (LINE) 64
SPOTWELD 183
SPRING FORWARD 245
SPRING ORIENTATION 249
SPRING/DAMPER 101
SUMMARY 268, 299
SUMMARY REPORT 46
SUPERPLASTIC FORMING 246
SURF NORMAL ON/OFF 73
SURFACE INTERSECT 72
SURFACE MESH 113
SURFACES 398
SWEEP SURFACE 76
SWITCH AXIS 310

T

TERMINATION 254
THERMAL MATERIAL MENU 216
THICK SHELL 100
TIE-BREAK 183
TIED NODES W/FAIL 183
TOP 363
TRACER 246
TRANSFORM - LINES 64
TRANSFORM - SURFACE 77
TRANSFORM - NODE 136
TRANSLATE 310
TRANSPARENCY FACTOR 380
TRANSPARENCY MODE ON/OFF 356
TRANSPARENT MODE 374
TRIM SURFACE 77
TRIM SURFACE - CIRCLE 78
TRIM SURFACE - DONE (CLOSED) 83
TRIM SURFACE - INTERSECTION LINE 79
TRIM SURFACE - OPEN CURVE 81
TRIM SURFACE - POINT 81
TRIM SURFACE - REJECT LAST POINT 83
TRIM SURFACE - SPLINE CURVE 81

TRUE VIEW 354

U

UTIL(ITY MENU) 361
UV LINE ON/OFF 72

V

VIEW PORTS ON/OFF 352
VIRTUAL AXES ROTATION 353

W

WRITE BINARY FILE 42
WRITE LINE DATA FILE 44
WRITE LS-DYNA FILE 43
WRITE MATERIAL 224
WRITE NASTRAN 45

Z

ZOOM 364